

**WL-TR-95-3060**

**Viscous Effects On  
Complex Configurations  
Software User's Manual**

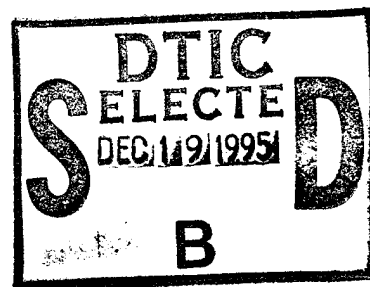
**Keith A. Burns  
Kenneth J. Deters  
CP Haley  
Thomas A. Kihlken**

**McDonnell Douglas Aerospace  
P.O. Box 516  
St. Louis, MO 63166**

**August 1995**

**Final Report for Period September 1992 - August 1995**

**Approved for public release; distribution is unlimited**



**19951218 093**

**Flight Dynamics Directorate  
Wright Laboratory  
Air Force Materiel Command  
Wright Patterson Air Force Base, Ohio 45433-7562**


**DTIC QUALITY INSPECTED 1**

## NOTICE

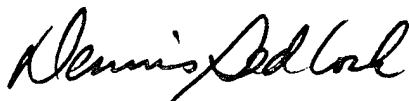
WHEN GOVERNMENT DRAWINGS, SPECIFICATIONS, OR OTHER DATA ARE USED FOR ANY PURPOSE OTHER THAN IN CONNECTION WITH A DEFINITE GOVERNMENT-RELATED PROCUREMENT, THE UNITED STATES GOVERNMENT INCURS NO RESPONSIBILITY OR ANY OBLIGATION WHATSOEVER. THE FACT THAT THE GOVERNMENT MAY HAVE FORMULATED OR IN ANY WAY SUPPLIED THE SAID DRAWINGS, SPECIFICATIONS, OR OTHER DATA, IS NOT TO BE REGARDED BY IMPLICATION, OR OTHERWISE IN ANY MANNER CONSTRUED, AS LICENSING THE HOLDER, OR ANY OTHER PERSON OR CORPORATION; OR AS CONVEYING ANY RIGHTS OR PERMISSION TO MANUFACTURE, USE, OR SELL ANY PATENTED INVENTION THAT MAY IN ANY WAY BE RELATED THERETO.

THIS REPORT IS RELEASABLE TO THE NATIONAL TECHNICAL INFORMATION SERVICE (NTIS). AT NTIS, IT WILL BE AVAILABLE TO THE GENERAL PUBLIC, INCLUDING FOREIGN NATIONS.

THIS TECHNICAL REPORT HAS BEEN REVIEWED AND IS APPROVED FOR PUBLICATION.

  
DONALD E. SHEREDA  
Aerospace Engineer  
Aero & Performance Group

  
RUSSEL F. OSBORN  
Technical Manager  
Aero & Performance Group

  
DENNIS SEDLOCK  
Chief,  
Aeromechanics Division

IF YOUR ADDRESS HAS CHANGED, IF YOU WISH TO BE REMOVED FROM OUR MAILING LIST, OR IF THE ADDRESSEE IS NO LONGER EMPLOYED BY YOUR ORGANIZATION PLEASE NOTIFY WL/FIMA, WRIGHT-PATTERSON AFB, OH 45433-7913 TO HELP MAINTAIN A CURRENT MAILING LIST.

COPIES OF THIS REPORT SHOULD NOT BE RETURNED UNLESS RETURN IS REQUIRED BY SECURITY CONSIDERATIONS, CONTRACTUAL OBLIGATIONS, OR NOTICE ON A SPECIFIC DOCUMENT.

<b>REPORT DOCUMENTATION PAGE</b>			<b>Form Approved</b> <b>OMB No. 0704-0188</b>	
Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington, DC 20503.				
1. AGENCY USE ONLY (Leave blank)		2. REPORT DATE August 1995		3. REPORT TYPE AND DATES COVERED Final Sept 1992 to August 1995
4. TITLE AND SUBTITLE Viscous Effects On Complex Configurations - Software User's Manual			5. FUNDING NUMBERS C: F33615-92-C-3005 PE: 62201 PR: 2404 TA: 07 WU: B9	
6. AUTHOR(S) K.A. Burns, K.J. Deters, CP Haley, T.A. Kihlken				
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) McDonnell Douglas Aerospace PO Box 516 St. Louis MO 63166-0516			8. PERFORMING ORGANIZATION REPORT NUMBER	
9. SPONSORING/MONITORING AGENCY NAMES(S) AND ADDRESS(ES) Flight Dynamics Directorate Wright Laboratory Air Force Materiel Command Wright-Patterson Air Force Base, Ohio 45433-7562			10. SPONSORING/MONITORING AGENCY REPORT NUMBER  WL-TR-95-3060	
11. SUPPLEMENTARY NOTES				
12a. DISTRIBUTION/AVAILABILITY STATEMENT  Approved for public release; distribution unlimited			12b. DISTRIBUTION CODE	
13. ABSTRACT (Maximum 200 words)  The Viscous Effects On Complex Configurations (VECC) program was a three task effort to improve preliminary aerodynamic prediction and analysis on high speed, arbitrary-shaped configurations. The first task was to incorporate a Newtonian streamline tracing code, "QUADSTREAM", into a Graphical User Interface (GUI) system. The second task consisted of upgrades to the Supersonic-Hypersonic Arbitrary Body Program (SHABP). The third task, Pre-/Post-Processing, encompassed the development of the geometry modeling/display module, SHABP pre-processor, and the line plotting program. The improvements to SHABP included: level 2 viscous analysis (including aeroheating), shock shape/flowfield analysis, control deflections, and trimmed aerodynamic predictions. The VECC program is documented in two reports, (1) the program final report, WL-TR-95-3059, (2) the software user's manual, WL-TR-95-3060.				
14. SUBJECT TERMS  Aerodynamics, High Speed, Hypersonic, Streamlines, Aeroheating			15. NUMBER OF PAGES 273	
			16. PRICE CODE	
17. SECURITY CLASSIFICATION OF REPORT  UNCLASSIFIED	18. SECURITY CLASSIFICATION OF THIS PAGE  UNCLASSIFIED	19. SECURITY CLASSIFICATION OF ABSTRACT  UNCLASSIFIED	20. LIMITATION OF ABSTRACT  SAR	

# TABLE OF CONTENTS

SECTION	PAGE
LIST OF FIGURES.....	ix
Preface .....	xiii
1. OVERVIEW .....	1-1
1.1 The VECC System.....	1-2
1.2 Graphical User Interface Design.....	1-4
1.2.1 Using Motif Widgets In The VECC GUI.....	1-5
1.2.2 VECC Main Window General Features .....	1-9
1.3 Main Window Menus.....	1-10
1.3.1 File Menu .....	1-11
1.3.1.1 Open .....	1-11
1.3.1.2 Save .....	1-12
1.3.1.3 Save As.....	1-12
1.3.1.4 Geometry Database .....	1-12
1.3.1.5 Add To Database.....	1-12
1.3.1.6 Print Geometry.....	1-12
1.3.1.7 Quit.....	1-13
1.3.2 Edit Menu.....	1-13
1.3.2.1 3D Geom Build .....	1-13
1.3.2.2 Panel Type .....	1-13
1.3.2.3 Panel Grouping .....	1-13
1.3.2.4 Input Units .....	1-13
1.3.2.5 Streamline Input.....	1-14
1.3.2.6 S/HABP Run Setup.....	1-14
1.3.2.7 S/HABP Case Setup.....	1-14
1.3.2.8 Trim Input.....	1-14
1.3.3 Display .....	1-14
1.3.4 Analysis.....	1-15
1.3.5 Results.....	1-15
1.3.6 Help.....	1-15
2. GEOMETRY MODELING .....	2-1
2.1 Geometry Modeling Conventions.....	2-2
2.1.1 Geometry Naming Conventions.....	2-2
2.1.2 Grid Point Orientation .....	2-3
2.1.3 Maximum Model Sizes .....	2-5
2.2 Model Display Modes .....	2-6
2.2.1 View Control.....	2-6

Dist		Avail and/or Special	
A-1		<input checked="" type="checkbox"/> <input type="checkbox"/> <input type="checkbox"/>	



## TABLE OF CONTENTS

SECTION	PAGE
2.2.2 Invisibility .....	2-7
2.2.3 Display Options .....	2-8
2.2.3.1 Symmetry .....	2-9
2.2.3.2 Opaque Panels .....	2-9
2.2.3.4 Show Orientation .....	2-9
2.2.3.5 Show Centers .....	2-10
2.2.3.6 Draw Triad .....	2-10
2.2.3.7 Draw Labels .....	2-10
2.2.3.8 Draw Normals .....	2-10
2.2.4 Grid Point Monitor .....	2-11
2.3 3D Geometry Build .....	2-11
2.3.1 Selecting Panels, Cross Sections or Grids .....	2-12
2.3.2 Geometry Editing Buttons .....	2-13
2.3.3 Geometry Editing Menus .....	2-14
2.3.3.1 By Panel .....	2-14
2.3.3.2 By Cross Section .....	2-14
2.3.3.3 By Grid .....	2-14
2.3.3.4 Delete .....	2-14
2.3.3.5 Copy .....	2-14
2.3.3.6 Flip Lateral .....	2-15
2.3.3.7 Aerosurface .....	2-15
2.3.3.8 Ellipse .....	2-15
2.3.3.9 Panel Library .....	2-15
2.3.3.10 Undo .....	2-15
2.3.3.11 Commit/Restore Geometry .....	2-15
2.3.4 Aerosurface Generator .....	2-15
2.3.5 Ellipse Builder .....	2-17
2.3.6 Geometry Part Library .....	2-20
2.4 Panel Grouping .....	2-21
2.5 Panel Type .....	2-21
2.6 Geometry Units .....	2-22
3. STREAMLINE ANALYSIS (QUADSTREAM) .....	3-1
3.1 QUADSTREAM Inputs .....	3-1
3.1.1 Part Definition .....	3-3
3.1.2 Flight Condition .....	3-3
3.1.3 Program Control Variables .....	3-4
3.1.3.1 Gap Tolerance .....	3-4
3.1.3.2 Symmetry .....	3-4
3.1.3.3 Streamline Direction .....	3-5

## TABLE OF CONTENTS

SECTION	PAGE
3.1.3.4 Streamline Definition .....	3-5
3.1.3.5 Additional Output .....	3-5
3.1.3.6 Streamline Starting/Stopping Conditions.....	3-6
3.1.4 Streamline Starting/Stopping Conditions .....	3-7
3.1.4.1 Distributed Starting Options.....	3-8
3.1.4.2 Element Starting Option.....	3-9
3.1.4.3 X,Y,Z Starting Option.....	3-10
3.1.4.4 Termination Options .....	3-10
3.2 Applicability for Complex Configurations .....	3-12
3.2.1 General Guidelines .....	3-12
3.2.2 Streamline Coverage.....	3-14
3.2.3 Premature Stopping of Streamlines.....	3-15
3.3 Error/Warning Messages .....	3-15
3.3.1 User Interface Error Message.....	3-15
3.3.2 I/O File Error Messages.....	3-18
3.3.3 Streamline Calculation Error Messages.....	3-18
3.4 Output Files .....	3-21
3.4.1 QUADSTREAM Default Streamline File (.qstr).....	3-22
3.4.2 VUAERO Streamline File (.qv#.##) .....	3-24
3.4.3 QUADSTREAM History File (.qhis).....	3-25
4. S/HABP ANALYSIS .....	4-1
4.1 Case Setup .....	4-1
4.1.1 Case Definition.....	4-2
4.1.1.1 Add/Copy/Delete Case .....	4-2
4.1.1.2 Edit Flight Conditions/Reference Quantities.....	4-3
4.1.1.3 Print Component Force Data .....	4-3
4.1.2 Summations.....	4-3
4.1.3 Select Other Runs To Be Executed .....	4-4
4.1.4 Flight Conditions/Reference Quantities.....	4-4
4.1.4.1 Reference Quantities .....	4-5
4.1.4.2 Derivatives.....	4-5
4.1.4.3 Flight Conditions .....	4-6
4.1.4.4 Flight Attitude.....	4-7
4.2 Run Setup .....	4-8
4.2.1 Run Definition.....	4-8
4.2.2 Analysis Type/Component.....	4-9
4.3 Flowfield/Shock Shape Analysis .....	4-10

## TABLE OF CONTENTS

SECTION	PAGE
4.3.1 General Inputs .....	4-10
4.3.2 Shock Expansion Inputs.....	4-13
4.3.2.1 General Shock Expansion Input .....	4-14
4.3.2.2 Cutting Plane Inputs.....	4-15
4.3.3 Surface Data Inputs .....	4-22
4.3.4 Uniform Flowfield Inputs .....	4-23
4.3.5 Non-Uniform Flowfield Inputs .....	4-23
4.3.6 Simple Flowfield Inputs .....	4-24
4.3.7 Limitations .....	4-25
4.3.8 Output .....	4-26
4.4 Inviscid Analysis.....	4-29
4.4.1 Pressure Method Selection.....	4-30
4.4.1.1 Windward Methods .....	4-30
4.4.1.2 Leeward Methods.....	4-32
4.4.2 Component Type .....	4-33
4.4.3 Control Deflection/Hinge Line .....	4-34
4.4.4 Apply Shielding Analysis .....	4-34
4.4.5 Apply Flowfield Interference.....	4-35
4.4.5.1 Hand Input Uniform Flowfield Data.....	4-37
4.4.5.2 Previously Calculated Flowfield Data .....	4-37
4.4.6 Use Stored Input Pressures .....	4-38
4.4.7 Detailed Print Flags .....	4-39
4.5 Level 1 Viscous Analysis .....	4-40
4.5.1 Level 1 Flags .....	4-42
4.5.2 Skin Friction Element Data.....	4-43
4.5.2.1 Element Number .....	4-44
4.5.2.2 Surface Element Area .....	4-44
4.5.2.3 Surface Element Initial Length.....	4-44
4.5.2.4 Surface Element Taper Ratio.....	4-45
4.5.2.5 Initial Surface Length.....	4-45
4.5.2.6 Initial Surface Taper Ratio .....	4-45
4.5.2.7 Wall Temperature .....	4-45
4.5.2.8 Flow Regime.....	4-45
4.5.3 Print Flags .....	4-46
4.5.3.1 Detailed Skin Friction Intermediate Results.....	4-46
4.5.3.2 Skin Friction Data For Each Element .....	4-46
4.5.3.3 Iteration or Final Local Cf Data .....	4-46
4.5.4 Inviscid Analysis For Input To Viscous Routines .....	4-46
4.6 Level 2 Viscous Analysis .....	4-47

## TABLE OF CONTENTS

SECTION	PAGE
4.6.1 Level 2 Viscous Input.....	4-48
4.6.1.1 Boundary Condition Input.....	4-49
4.6.1.2 Radius of Influence .....	4-50
4.6.1.3 Gas Equation Flag.....	4-50
4.6.1.4 Boundary Layer State.....	4-51
4.6.1.5 Transition Criteria.....	4-51
4.6.1.6 Boundary Layer Model .....	4-52
4.6.1.7 Flat Plate.....	4-53
4.6.1.8 Leading Edge .....	4-54
4.6.1.9 Inviscid Analysis for Input to Viscous Routines .....	4-56
4.6.1.10 Separation Criteria .....	4-56
4.6.1.11 Print Flag .....	4-57
4.6.2 Output .....	4-57
4.6.2.1 Text Output.....	4-58
4.6.2.2 Graphical Output.....	4-59
5 TRIM ANALYSIS .....	5-1
5.1 Trim Inputs .....	5-1
5.1.1 Case Selection .....	5-2
5.1.2 Summation/Deflection Input.....	5-2
5.1.3 Extrapolation Flag .....	5-3
5.1.4 Center of Gravity Input.....	5-3
5.2 Trimmed Aerodynamic Output .....	5-3
6. RESULTS DISPLAY .....	6-1
6.1 Line Plotting .....	6-1
6.1.1 Hplot User Interface .....	6-3
6.1.1.1 Menus .....	6-3
6.1.1.2 Dialog Boxes.....	6-3
6.1.2 Data Window.....	6-4
6.1.2.1 Data Layers.....	6-4
6.1.2.2 Editing a Data Table .....	6-4
6.1.2.3 Data Window Menus.....	6-5
6.1.3 Hplot Windows.....	6-10
6.1.3.1 Tools Menu.....	6-11
6.1.4 Print Graphs .....	6-15
6.1.5 Hplot Plotting Limitations .....	6-15
6.2 View Streamlines .....	6-17
7. REFERENCES.....	7-1

## TABLE OF CONTENTS

SECTION	PAGE
APPENDIX A - INPUT DATA INSTRUCTIONS	
APPENDIX B - <i>PREPMK5 (Text Preprocessor)</i> INSTRUCTIONS	

## LIST OF FIGURES

FIGURE	TITLE	PAGE
1.1-1	VECC System Architecture .....	1-3
1.1-2	VECC File Summary.....	1-4
1.2-1	Main Window Basic Characteristics .....	1-5
1.2-2	Windows Can Be Divided Into Different Panes .....	1-6
1.2-3	Sample Scroll Bar .....	1-6
1.2-4	Sample Buttons .....	1-7
1.2-5	Sample Option Menu.....	1-7
1.2-6	Example Of Editing A Text Field In A Dialogue Box .....	1-8
1.2-7	Example Of Making Selection In Dialogue Boxes.....	1-8
1.2-8	Example Of Window With OK, APPLY, CANCEL and HELP Buttons.....	1-9
1.2-9	VECC GUI Main Window .....	1-10
1.3-1	File Menu.....	1-11
1.3-2	Example Of Open File Window .....	1-12
1.3-3	Edit Menu Selections.....	1-13
1.3-4	Display Menu .....	1-14
1.3-5	Analysis Menu .....	1-15
1.3-6	Results Menu.....	1-15
1.3-7	Help Menu .....	1-16
2.1-1	Geometry Definitions .....	2-3
2.1-2	Point Numbering for Body .....	2-4
2.1-3	Point Numbering for Aerosurfaces .....	2-5
2.1-4	Point Numbering for Vertical Tails .....	2-5
2.2-1	View Control Buttons.....	2-6
2.2-2	Invisibility Menu Options.....	2-7
2.2-3	Display Menu Options.....	2-9
2.2-4	Modeler Display with Triad and Labels.....	2-10
2.3-1	3D Build Editing Menus.....	2-12
2.3-2	Geometry Editing Buttons .....	2-13
2.3-3	Aerosurface Generator Window .....	2-16
2.3-4	Input Definition For Airfoil Shapes .....	2-17
2.3-5	Ellipse Builder Window .....	2-18
2.3-6	Sample Cross Sections Modeled With The Ellipse Builder .....	2-19
2.3-7	Nose Shape Options .....	2-20
2.5-1	Panel Type Window .....	2-22
2.6-1	Input Units Submenu.....	2-23
3.1-1	QUADSTREAM Input Window.....	3-2
3.1-2	Part Definition Inputs .....	3-3
3.1-3	Flight Conditions Input.....	3-4
3.1-4	Geometry Gap Tolerance Input .....	3-4
3.1-5	Geometry Symmetry Option.....	3-4
3.1-6	Streamline Direction Option.....	3-5
3.1-7	Streamline Definition Options .....	3-5
3.1-8	Streamline Output Options .....	3-6
3.1-9	Streamline Starting/Stopping Options.....	3-6
3.1-10	Streamline Starting/Stopping Conditions Window.....	3-7

## LIST OF FIGURES

FIGURE	TITLE	PAGE
3.1-11	Distributed Starting Options .....	3-9
3.1-12	Element Starting Option .....	3-9
3.1-13	X,Y,Z Starting Option .....	3-10
3.1-14	Part Tracing Option .....	3-10
3.1-15	Streamline Termination Option .....	3-11
3.1-16	Streamline Termination Option For Wing, Fin And Tail Parts .....	3-11
3.2-1	Surface Versus Quadrilateral Representation .....	3-13
3.2-2	Streamline Coverage At Angle-Of-Attack. ....	3-14
3.3-1	QUADSTREAM Error Dialog .....	3-15
3.4-1	QUADSTREAM Generated Streamlines On AFWAL Elliptic Body (Alpha = 0.0).....	3-21
3.4-2	QUADSTREAM Default Output File Format.....	3-23
3.4-3	QUADSTREAM Default Output File For AFWAL Elliptic Body. ....	3-23
3.4-4	VUAERO Streamline File Format. ....	3-24
3.4-5	VUAERO Streamline File For AFWAL Elliptic Body (Alpha = 0.0). ....	3-24
3.4-6	QUADSTREAM History File For AFWAL Elliptic Body.....	3-25
4.1-1	Case Input Window .....	4-2
4.1-2	Flight Conditions Window .....	4-4
4.1-3	Available Derivative Options .....	4-5
4.1-4	Derivatives Calculated.....	4-5
4.1-5	Flight Condition Input.....	4-7
4.1-6	Flight Attitude Input.....	4-7
4.1-7	Any Flight Attitude Variable Can Be Input As A Table.....	4-8
4.2-1	Run Definition Window .....	4-9
4.2-2	Analysis Type/Component .....	4-9
4.3-1	Flowfield Analysis Window .....	4-11
4.3-2	Flowfield Analysis Type .....	4-12
4.3-3	Access To Flowfield Analysis Inputs.....	4-12
4.3-4	Detailed Flowfield Output Option .....	4-12
4.3-5	VUAERO Results File Inputs.....	4-13
4.3-6	Shock Expansion Input Window.....	4-13
4.3-7	Shock Expansion Order.....	4-14
4.3-8	Flowline Source Option Menu.....	4-14
4.3-9	Shock Expansion Starting Solution Menu.....	4-15
4.3-10	Body Slope Option Menu .....	4-15
4.3-11	Shock Expansion Output Options .....	4-15
4.3-12	Blunt Nose Starting Inputs.....	4-15
4.3-13	Cutting Plane Axis Orientation To Reference Axis System.....	4-16
4.3-14	Meridian Plane Orientation.....	4-17
4.3-15	Meridian Plane Example .....	4-17
4.3-16	Parallel Cutting Planes .....	4-18
4.3-17	Number Of Cutting Planes.....	4-19
4.3-18	Cutting Plane Type Option Menu .....	4-19
4.3-19	Cutting Plane Spacing Option Menu.....	4-19
4.3-20	Cutting Plane Surface Normal Option Menu.....	4-20

## LIST OF FIGURES

FIGURE	TITLE	PAGE
4.3-21	Cutting Plane Detailed Print Option .....	4-20
4.3-22	Cutting Plane Orientation Input .....	4-20
4.3-23	Input Meridian Cutting Plane Spacing .....	4-21
4.3-24	Input Parallel Cutting Plane Spacing .....	4-21
4.3-25	Hand Input Surface Data Window .....	4-22
4.3-26	Uniform Flowfield Inputs .....	4-23
4.3-27	Non-Uniform Flowfield Data Window .....	4-24
4.3-28	Simple Flowfield Inputs .....	4-25
4.3-29	Detailed Cutting Plane Output .....	4-27
4.3-30	Detailed Shock Expansion Output .....	4-28
4.3-31	Detailed Flowfield Output .....	4-28
4.4-1	Inviscid Pressure Input Window .....	4-29
4.4-2	Windward Inviscid Pressure Methods .....	4-30
4.4-3	Leeward Inviscid Pressure Methods .....	4-32
4.4-4	Component Type Option Menu .....	4-34
4.4-5	Shielding Analysis Input Window .....	4-35
4.4-6	Flowfield Interference Window .....	4-37
4.4-7	Input Pressure Option Window .....	4-39
4.5-1	Example Of Viscous Geometry Compared To Inviscid Geometry .....	4-41
4.5-2	Level 1 Viscous Analysis Window .....	4-42
4.5-3	Level 1 Viscous Flags .....	4-43
4.5-4	Skin Friction Method Options .....	4-43
4.5-5	Skin Friction Element Input .....	4-44
4.5-6	Initial Surface Taper Ratio Example .....	4-45
4.5-7	Skin Friction Optional Output .....	4-46
4.6-1	Streamline Coverage Affects Accuracy of Level 2 Analysis .....	4-48
4.6-2	Level 2 Viscous Analysis Window Specifies Input per Component .....	4-49
4.6-3	Boundary Condition Inputs .....	4-49
4.6-4	Input Box for the Radius of Influence Parameter .....	4-50
4.6-5	Gas Models Available in Level 2 Viscous Analysis .....	4-51
4.6-6	Level 2 Viscous Analysis Includes Transition Capability .....	4-51
4.6-7	Transition Criteria Options .....	4-52
4.6-8	Component Modeled as Leading Edge or Flat Plate .....	4-53
4.6-9	Inputs Associated with Flat Plate Methods .....	4-53
4.6-10	Turbulent Flat Plate Boundary Layer and Aeroheating Methods .....	4-53
4.6-11	Cone Correction for Flat Plate Analysis .....	4-54
4.6-12	Inputs Associated with Leading Edge Method .....	4-54
4.6-13	Leading Edge Types Available in Level 2 Viscous analysis .....	4-55
4.6-14	Swept Cylinder Analysis Methods .....	4-55
4.6-15	Inviscid Analysis Input for Level 2 Viscous Analysis .....	4-56
4.6-16	Flow Separation Criteria Menu .....	4-57
4.6-17	Level 2 Viscous Analysis Output Formats .....	4-57
4.6-18	Streamline Result Output Format .....	4-58
4.6-19	Interpolated Element Results Format .....	4-59
5.1-1	Trim Option Window .....	5-2



## LIST OF FIGURES

FIGURE	TITLE	PAGE
5.2-1	Sample Trim Output File.....	5-4
6.1-1	Hplot Line Plotting Program Invoked from VECC.....	6-2
6.1-2	Sample Plot from a Customized Quickplot.....	6-10
6.1-3	Effect of Smoothing Data Plots.....	6-13
6.2-1	Display, Menu and Dialog box Used for Streamline Viewing.....	6-17

## Preface

This report was prepared for Wright Laboratory (WL), Wright-Patterson AFB, Ohio under Contract Number F33615-92-C-3005. The work was performed by McDonnell Douglas Aerospace (MDA), St. Louis, Missouri. Mr. Don Shereda (WL/FIMA) was the USAF Project Engineer and Mr. Keith Burns was the MDA Program Manager. Mr. Ken Deters, Mr. CP Haley, and Mr. Tom Kihlken were the principal investigators and program developers for this contract.

The Viscous Effects On Complex Configurations (VECC) contract was undertaken to update the Supersonic/Hypersonic Arbitrary Body Program (S/HABP) and to develop a complete analysis environment using a graphical user interface. This contract has lead to a new release of S/HABP designated S/HABP Mark V which encompasses the best method improvements developed under the earlier Missile Aerodynamic Design Methods (MADM, contract F33615-84-C-3002), and Stability and Control Of Hypersonic Vehicles (SCHV F33615-86-C-3602) programs. In addition, the capability to analyze aeroheating was added to S/HABP. The Graphical User Interface (GUI) incorporates modules for geometry display/build, streamline tracing (developed under the Missile Supersonic/Hypersonic Aerodynamics contract F33615-89-C-3003), S/HABP pre/post processor, and aerodynamic trimming.

The VECC contract contained three main tasks; (1) QUADSTREAM, (2) S/HABP Upgrades, and (3) Pre-/Post Processing. The first task, QUADSTREAM, involved the incorporation of the QUADSTREAM code which traces streamlines on quadrilateral based geometries into the GUI. The second task, S/HABP upgrades, primarily included improvements to the level 2 analysis methods and formal incorporation of methods added to S/HABP Mark IV after the Mod 0 release. The improvements to S/HABP included: level 2 viscous analysis (including aeroheating), shock shape/flowfield analysis, control deflections, and trimmed aerodynamic predictions. The trimmed aerodynamic prediction method was incorporated into the GUI as a post-processor rather than into S/HABP itself. The third task, Pre-/Post Processing, encompassed the development of the geometry modeling/display module, S/HABP preprocessor, and the line plotting program. Each of these capabilities was incorporated into the module referred to as the Graphical User Interface (GUI). In addition, a text based preprocessor was developed for working with non X Windows terminals.

The authors would like to extend their appreciation to Mr. Don Shereda and Mr. Val Dahlem for their contributions and guidance during the development of this program. In addition, the authors would like to recognize the valuable input received from the following individuals Mr. Brian Bennett, Mr. Joe Gregiore, Mr. Andy Jenn, and Mr. Kei Lau.

## 1. OVERVIEW

The Viscous Effects On Complex Configurations (VECC) program had the objective to provide essential aerodynamic prediction capabilities necessary to assess the performance characteristics of advanced vehicles and combine these capabilities with pre- and post processors to create an aerodynamic analysis system. Three primary tasks aimed at improving the accuracy, versatility, and ease of use of the Supersonic/Hypersonic Arbitrary Body Program (S/HABP, Reference 1) were completed under this effort and a new version of S/HABP has been released designated S/HABP Mark V. In addition to the new version of S/HABP a Graphical User Interface (GUI) was developed which creates a complete analysis environment. The GUI incorporates the capability to build a geometry, trace streamlines on the quadrilateral geometry, display streamlines on the geometry, prepare input to S/HABP Mark V, run S/HABP Mark V, trim the S/HABP Mark V results, and plot the results in report quality format. This document is a User's Manual for S/HABP Mark V and provides the necessary information for using the VECC analysis system. The Final Report contains details of the program development as well as validation of the new and updated methods.

S/HABP Mark V contains new or updated methods in the following areas: (1) Stability Derivatives, (2) Control Surface Deflections, (3) Flowfield/Shock Shape, (4) Boundary Layer/Aeroheating. Stability derivatives were not part of the official release of S/HABP Mark IV, Mod 0, but were incorporated separately at a later date. Under this effort, the stability derivative methods were validated and new plunge derivative methods added. Control surface deflection methodology was updated to allow multiple control deflections in a single case and created a separate definition for roll/yaw deflections. New methods were added and input greatly simplified for predicting the shock shape generated by a component and analyzing the resulting flowfield. A new level 2 viscous methodology was added which takes advantage of the new quadrilateral streamline tracing capability and includes aerodynamic heating.

The challenge in developing the GUI was to encompass the truly arbitrary nature of S/HABP input files while providing sufficient structure to the program to decrease analysis set up time. To this end, compromises were made between ease of use and versatility. The user is cautioned that although the GUI makes input easier, he/she should read this manual to become familiar with the flow of the input file generation. Users of S/HABP Mark IV will find that several capabilities have been combined into single windows. For example in Mark IV to perform a level 1 viscous analysis, a prior inviscid pressure run must have been input by the user and the pressure data saved before executing the viscous run input. In Mark V, the same sequence of runs are performed but they are transparent to the user, and the user enters both the viscous input and the matching inviscid input in the same window.

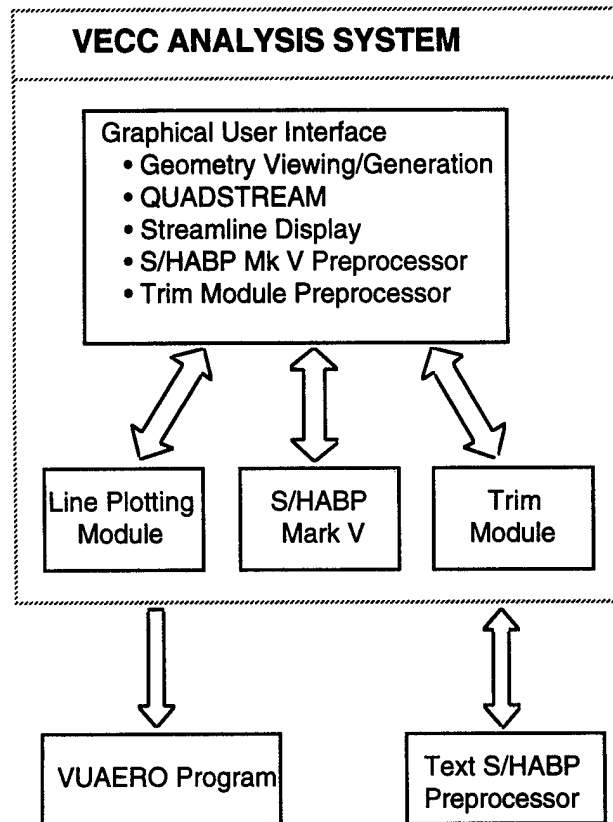
The remainder of Section 1 will discuss the overall architecture of the new analysis system and some general philosophy used in organizing the GUI. Users new to the graphical workstation environment are strongly encouraged to read Section 1.2 which provides general data on using Motif widgets (buttons, slides, etc.).

## **1.1 The VECC System**

The VECC system consists of four modules as shown in Figure 1.1-1. The primary module is the Graphical User Interface (GUI) which is used to view/generate geometry, view/generate streamlines, generate input for S/HABP Mark V and trim program, execute S/HABP and the trim program, and launch the line plotting program. The Line Plotting Program, S/HABP Mark V, and the Trim Module are separate programs that are tied directly to the GUI (i.e., can be executed from the GUI with file transfer transparent to the user). The VUAERO program (a separate code developed under the Missile Supersonic/Hypersonic Aerodynamics contract, Reference 2 and 3) and the Text Preprocessor are not an integrated part of the VECC system, but share files with it.

The file structure is summarized in Figure 1.1-2 with a description of the contents of each file. The files marked with an asterisk are created for each execution of the program (i.e., are standard input/output), and the other files are optional input/output files.

The graphical user interface is designed to take advantage of the graphics available on Unix based workstations. The selected graphical environment uses the X Window/Motif standard libraries. X Windows has become an industry standard windowing system and is available for virtually all computers running a Unix based operating system. Motif provides a standard package of X Window "widgets and gadgets" that make application development faster and more uniform. The VECC GUI was developed using the Motif style guide (Reference 4).



**Figure 1.1-1 VECC System Architecture**

File	Description
file.mk5 *	S/HABP Mark V input file
file.out *	S/HABP Mark V output file
file.geo	Geometry file (Type 3 Cards)
file.qstr	QUADSTREAM streamline file
file.qvu.N	Streamline traces in VUAERO display format. Final extension N is an integer 1-99 with each number corresponding to the $\alpha$ - $\beta$ sequence run in the GUI.
file.qhis	Streamline history file (element by element trace and warning messages)
file.cp	Inviscid pressure coefficient data for display in VUAERO
file.trm	Trim aero output file
SHIELD.DAT.N	VUAERO neutral file containing shielded elements as calculated in the shielding program. Each "panel" in this file contains all shielded elements for one $\alpha$ - $\beta$ combination. The extension N is incremented for each case in which shielding output is requested.
FFIELD.DAT.N	Flowfield/shock shape file for display in VUAERO. This file always has the same name with the final extension N being a number from 1 to 99. A new file is created for each $\alpha$ - $\beta$ sequence run in the GUI.
VISCOUS.ERF.N	Element Results File for input to VUAERO. This file always has the same name with the final extension N being a number from 1 to 99. A new file is created for each call to the level 2 viscous routines (i.e., each case analyzed).
VISCOUS.TP3.N	Type 3 geometry cards corresponding to the level 2 viscous analysis. Used in conjunction with VISCOUS.ERF.N for VUAERO display.
POSTSC.DAT	Stability and control postprocessor aerodynamic input file. See Auxiliary Program Input, Appendix A of the Mark V User's Manual.

**Figure 1.1-2 VECC File Summary**

## 1.2 Graphical User Interface Design

This section discusses the GUI design used for the VECC system and the use of Motif widgets in general. Users who are conversant with the use of slides, radio buttons, option buttons, and pop-up menus can skip to Section 1.2.2 which discusses the general window layout or Section 1.2.3 which discusses the main GUI window in detail.

### 1.2.1 Using Motif Widgets In The VECC GUI

This section focuses on the basic window operations found in virtually all Motif programs. The user can significantly decrease his/her learning time and frustration level by reviewing these basic operations.

Upon opening the GUI there are several general characteristics of the window you should notice. First notice the window frame as shown in Figure 1.2-1. The window frame will appear slightly different depending on the exact window manager your operating system has, but it will have the same basic features as shown. Just below the Title Bar is the Menu Bar which will appear only in the main window and contains a series of pull down menus. The main window menus are discussed in more detail in Section 1.3. Windows are sometimes divided into separate window panes. This is often done for improved visual organization of the window. The main VECC GUI window is divided into four window panes. The vertical size of the panes can be adjusted via a button as shown in Figure 1.2-2. Section 1.2.2 contains a more detailed discussion of the operations which can be performed in the main window.

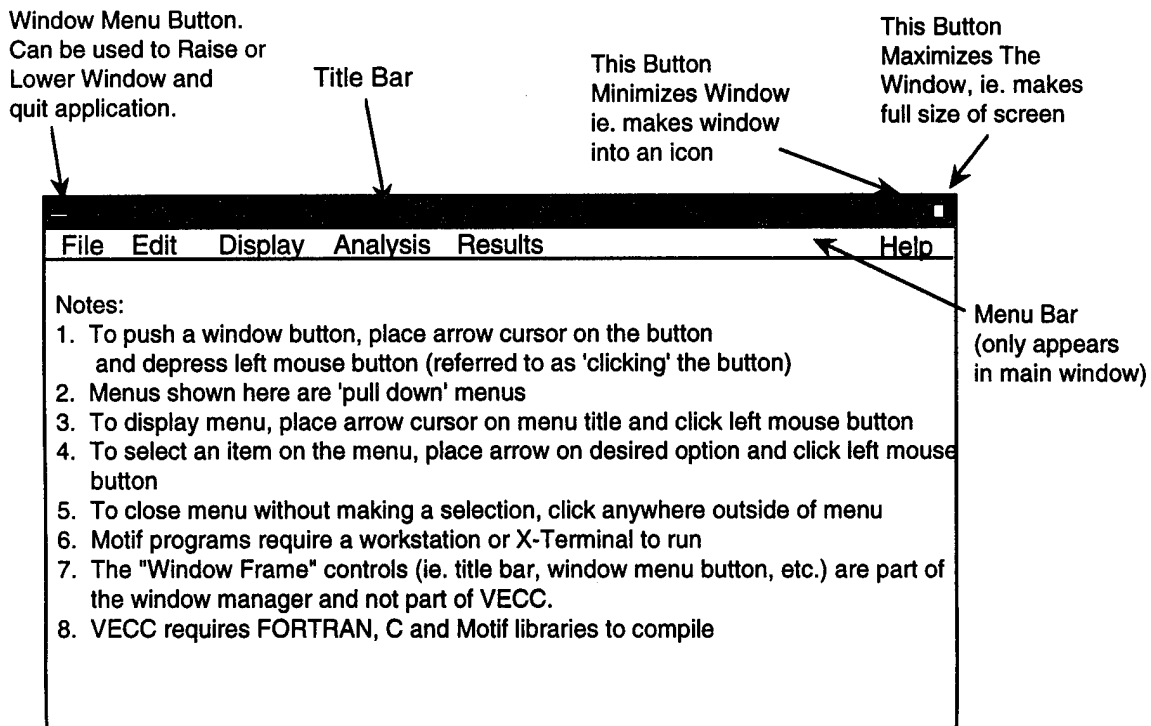
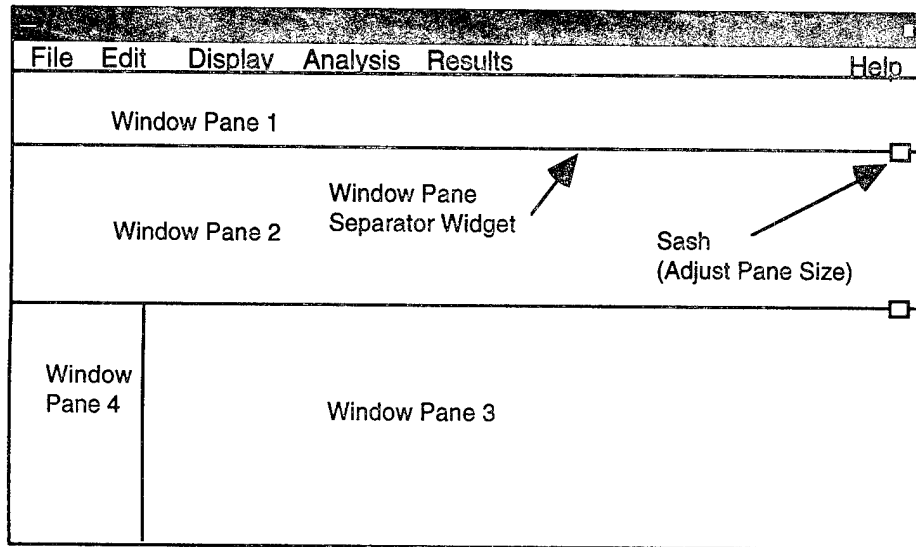
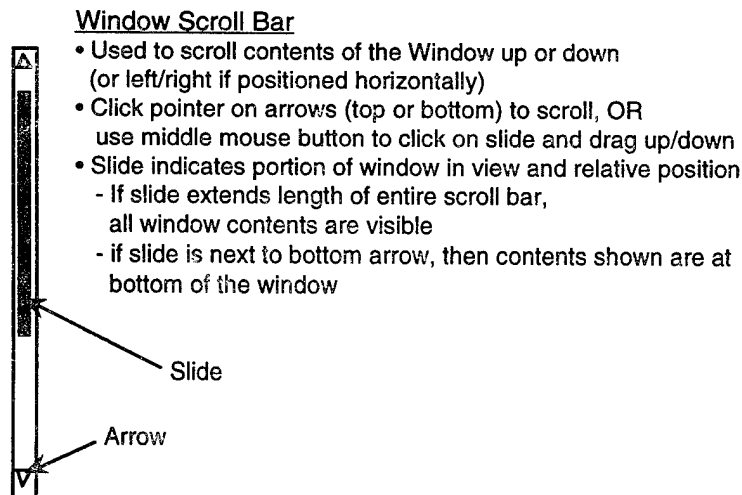


Figure 1.2-1 Main Window Basic Characteristics



**Figure 1.2-2 Windows Can Be Divided Into Different Panes**

Many windows contain scroll bars which are used to adjust the viewing area inside of the window or a text dialogue box. An example of a scroll bar is shown below in Figure 1.2-3



**Figure 1.2-3 Sample Scroll Bar**

There are two styles of buttons used in the VECC GUI. The first is a plain button which is square. When the button is depressed, the option listed next to the button has been selected. The second type of button is referred to as a radio button. This button appears as a diamond, and is exclusive. That is, when one radio button is depressed all of the others pop out. This type of button is used when one and only one selection can be made. Example buttons in a dialogue area are shown in Figure 1.2-4. A third type of button, the text button, is often used in the

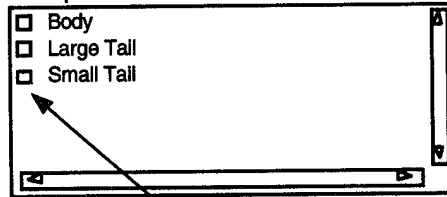


VECC GUI. Text buttons appear as plain labels with the exception that they are followed by three dots (...) and are pushed simply by clicking on any portion of the text.

#### Buttons

- Either square or diamond shape
- Selection is active when button is pushed in
- Diamond shaped buttons (referred to as radic buttons) are exclusive i.e., pushing one button in pops other buttons out

#### Components



Example of a 'button'


**Figure 1.2-4 Sample Buttons**

Option menus are used extensively in the GUI. Option menus provide the user with a list of available choices and allows the user to make a selection. An example of an option menu is shown in Figure 1.2-5.

#### Option Menu


- These menus list available options for a given input field as shown below
- To view available options, click in menu field and options will pop up
- To select an option, click on the desired option with menu open

#### Option Menu (Closed)

Impact Pressure Method: **Dahlem-Buck** 

Click and hold down the left mouse button and the menu stays open as long as mouse button is depressed.

#### Option Menu (Open)

Impact Pressure Method: 

Tangent Cone
Tangent Wedge
Dahlem-Buck
Newtonian
Mod. Newtonian

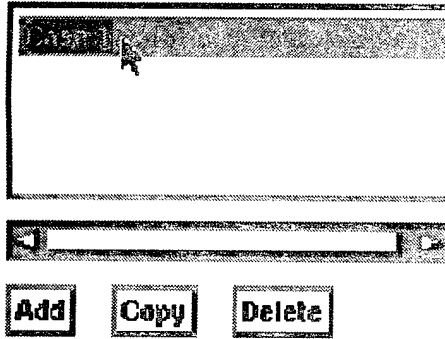
Click on the option menu and it remains open until a selection is made.

**Figure 1.2-5 Sample Option Menu**

Dialogue boxes are places where the user can enter or select text fields. The user enters data by clicking in the field and typing. An entire word or number can be selected by double clicking the left mouse button. An entire line can be selected by triple clicking the left mouse button. On certain dialogues such as the summation dialogue, the user may desire to select more than one text field. This can be done in one of two ways. If the user wants to select a range of text fields,

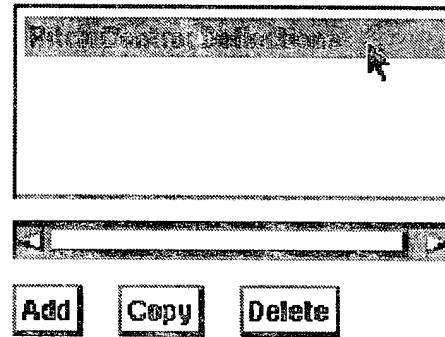
he/she can click on the first text field and then holding down the shift key, click on the last text field. Thus all fields between the first and last are selected. If the user wants to select text fields which are not in sequence, he/she should hold down the control key while clicking on the desired selection. Examples follow in Figures 1.2-6 and 1.2-7.

#### Case Definition



- The text field has been selected by clicking and dragging the cursor over the text.

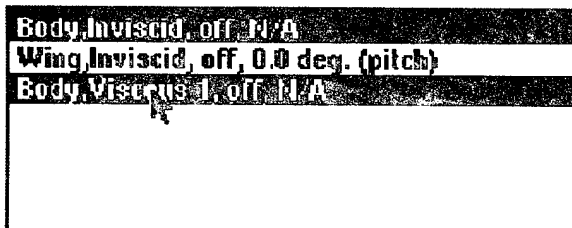
#### Case Definition



- After using the delete key to remove the original text, a new case title is entered.

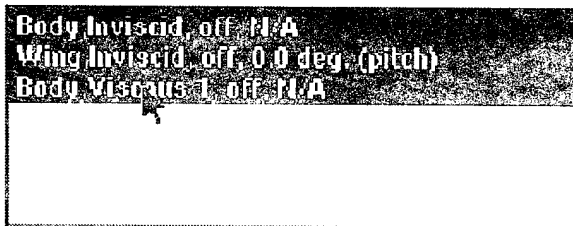
Figure 1.2-6 Example Of Editing A Text Field In A Dialogue Box

#### Select Runs for this Summation



- Select two runs for summation
- Click on the first run
- Run is highlighted to show it has been selected
- Hold down control key
- Click on the third run.
- First and third runs are selected

#### Select Runs for this Summation

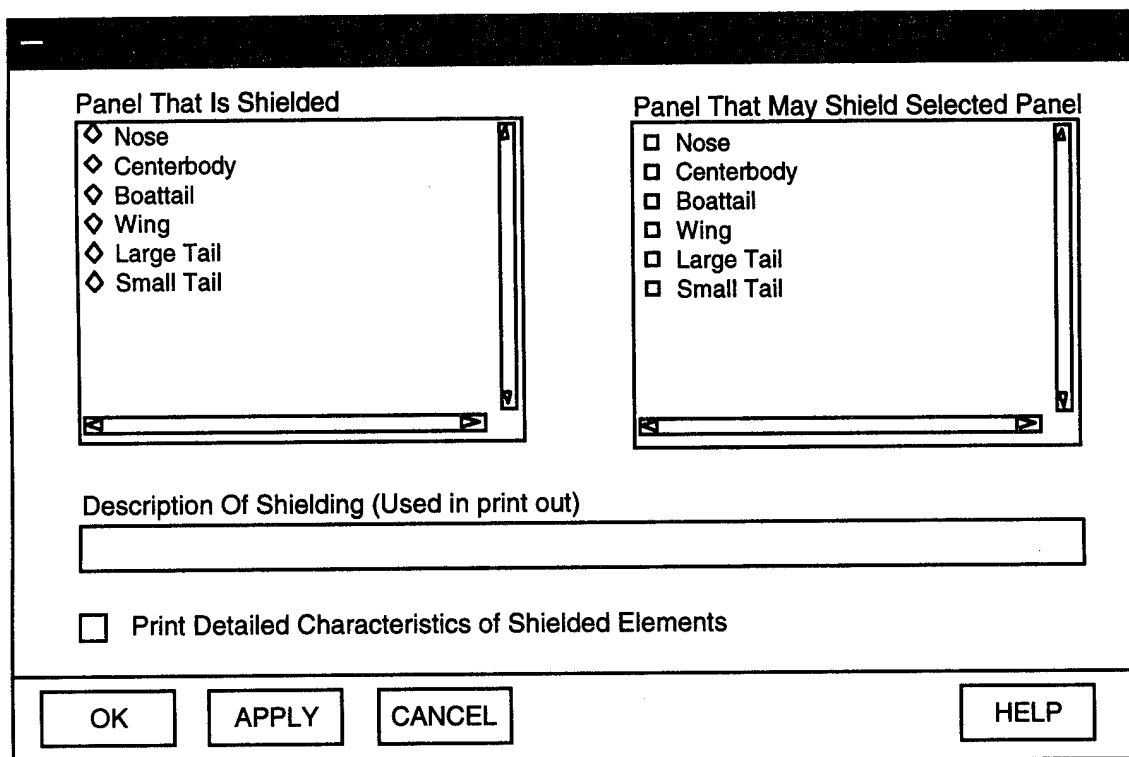


- Select an entire group
- Click on first selection
- Hold down shift key
- Click on last selection
- Entire range now selected

Figure 1.2-7 Example Of Making Selection In Dialogue Boxes

Most of the VECC GUI windows will have four buttons at the bottom. These buttons are labeled "OK," "Apply," "Cancel" and "Help." The OK button will accept the input created in this window, close the window and return to the previous window. The CANCEL button voids

the input created in this window and returns to the previous window. The APPLY button makes the input generated in this window permanent, runs any available input error checking, and then remains in the present window. The HELP button will provide on-line help for the window being displayed. The on-line help windows are editable text fields that the user may edit, add or delete within.



**Figure 1.2-8 Example Of Window With OK, APPLY, CANCEL and HELP Buttons**

### **1.2.2 VECC Main Window General Features**

The main window of the VECC GUI as shown in Figure 1.2-9 is divided into three main areas via adjustable window panes. The top window pane is known as the feedback window. Messages and warnings from the GUI and S/HABP are shown here. The middle window pane contains edit and display functions for geometry manipulation. The functions of this window pane are discussed more fully in Section 2.2 and 2.3. The bottom window pane contains the geometry build and display functions. On the left are menus for building and modifying the geometry. On the right is the geometry display area. This window pane is discussed further in Sections 2.4 and 2.5.

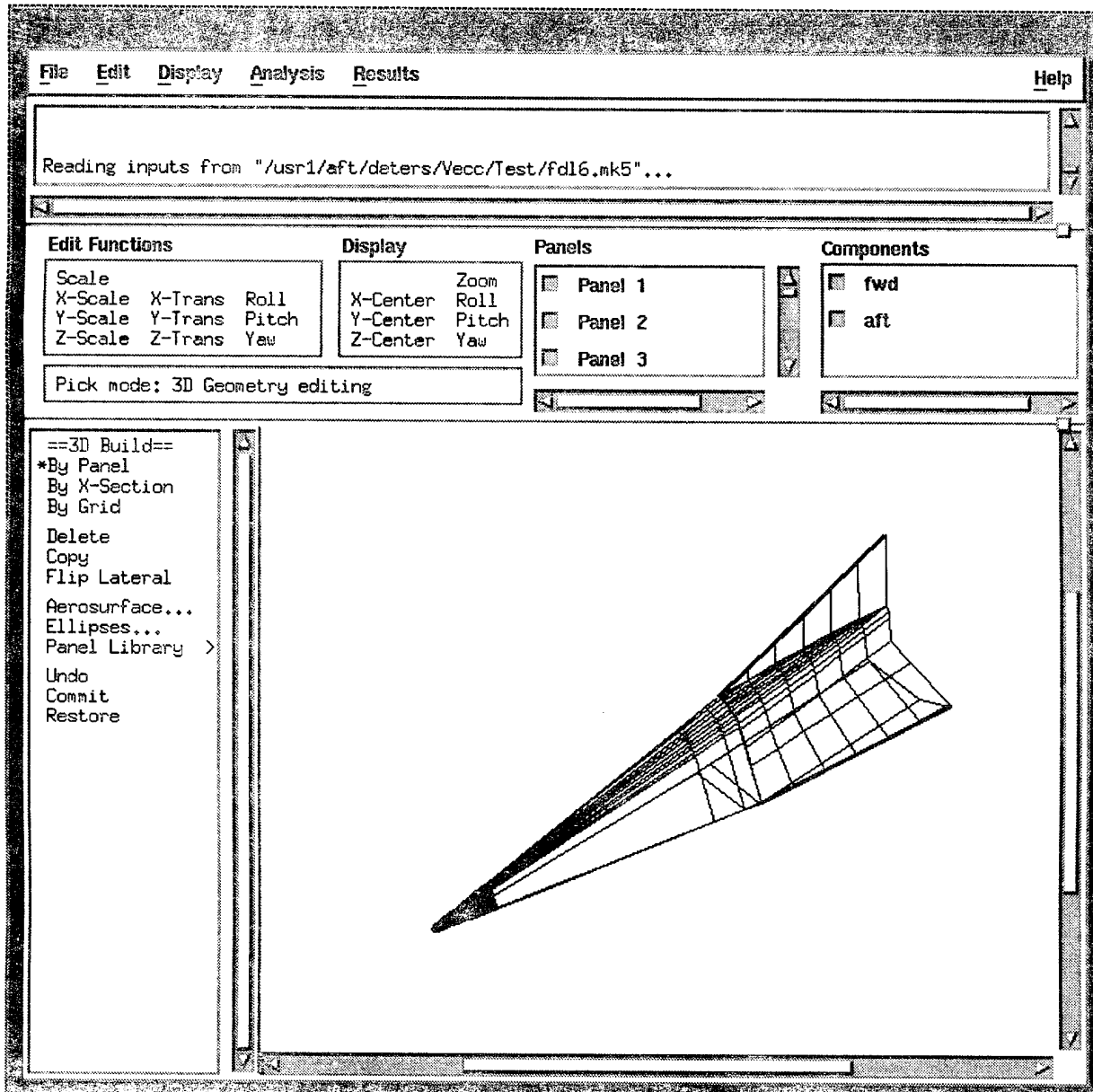


Figure 1.2-9 VECC GUI Main Window

### 1.3 Main Window Menus

The main window menus are "File," "Edit," "Display," "Analysis," "Results," and "Help." These provide the primary access of the user to the VECC system capabilities. The three dots (...) appearing after the menu items indicate that after making the selection a dialogue window will be displayed for further input. When a triangle appears after a menu option (such as **Input Units ▾**), a submenu will pop up if this item is selected. Each main window menu is described below.

### 1.3.1 File Menu

The file menu is used for file input and output operations. The menu options are shown below in Figure 1.3-1.

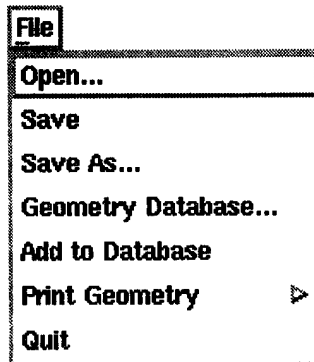
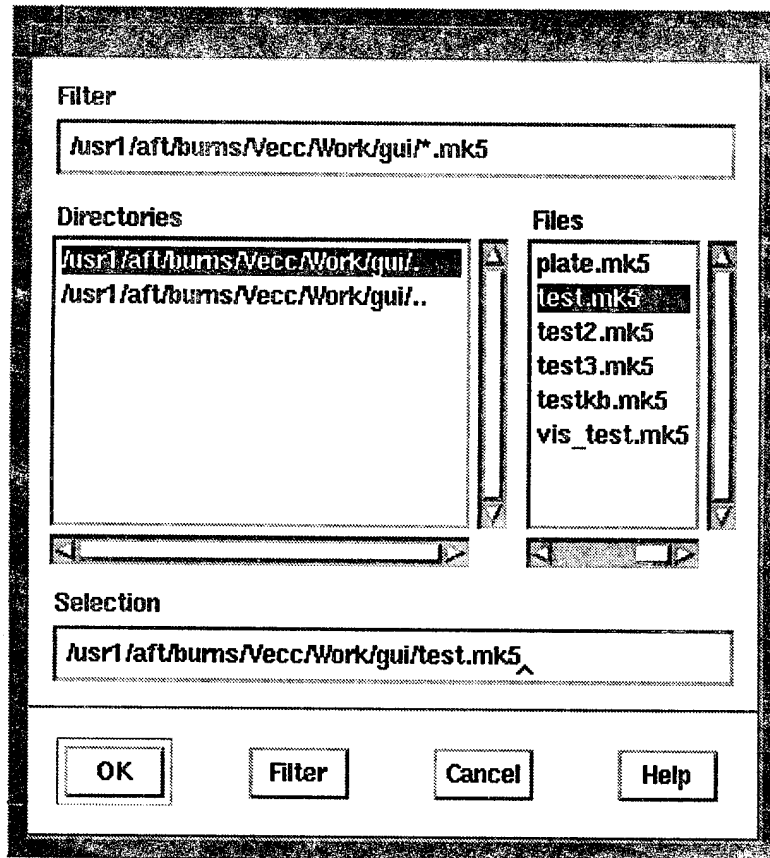


Figure 1.3-1 File Menu

**1.3.1.1 Open** - Selecting this option launches the open file window shown below in Figure 1.3-2. Either a ".mk5" file or a ".geo" can be read into the GUI. The dialogue area labeled "Filter" tells the program from what directory and which type of files to list in the "Files" dialogue. The "Directories" dialogue list lists any directories available in the current directory. At a minimum the "Directories" dialogue list will contain a "/" and a "/" directory. The "/" represents the current directory, and "/" is for the directory one directory level above the current directory. You can double click on a file listed in the "Files" dialogue to open the file or type a file name in the "Selection" window.



**Figure 1.3-2 Example Of Open File Window**

**1.3.1.2 Save** - Selecting "Save" from the "File" menu saves the ".mk5" input file which contains all of the S/HABP, geometry and GUI input. If a new file is being created, selecting "Save" opens the "Save As ..." window which is discussed below.

**1.3.1.3 Save As** - Selecting "Save As ..." will open a window which allows the user to enter a new name for the ".mk5" file. This window is essentially the same as the "Open File" window.

**1.3.1.4 Geometry Database** - Selecting "Geometry Database ..." opens a window where the user specifies a geometry library name. The geometry library can be used for extracting "pre-made" panels which may be used as is or modified.

**1.3.1.5 Add To Database** - This file menu option allows the user to add the selected geometry panels to the library previously opened with the "Geometry Database ..." command.

**1.3.1.6 Print Geometry** - This selection allows the user to send output of the geometry as currently displayed to a postscript printer in either portrait or landscape format.

**1.3.1.7 Quit** - Selecting this menu option closes the program. If the user has made changes in the GUI without saving, he/she will be prompted, warned and given the opportunity to save the file before exiting the program. If the changes are saved, the previous ".mk5" file is saved as ".bck"

### 1.3.2 Edit Menu

This menu controls what type of input the user is providing. The first four selections relate to the building of the geometry. The remaining selections launch windows for input to the streamline tracing program, S/HABP, and the trim program. Each of these options is discussed below. The options under the "Edit Menu" are shown in Figure 1.3-3.

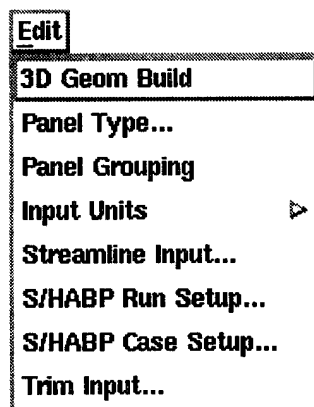


Figure 1.3-3 Edit Menu Selections

**1.3.2.1 3D Geom Build** - This menu selection puts the GUI in 3D Build mode (default upon entering the GUI). This causes the "3D Build" menu to appear in the lower left window pane. The operations available while in this mode are discussed in detail in Section 2.4.

**1.3.2.2 Panel Type** - This menu selection launches the "Panel Type" window shown in Figure 2.5.1. In this window the user specifies attributes of each of the panels. These attributes include cross section orientation, panel symmetry, and whether the panel is "inviscid" or "viscous." This window is discussed in detail in Section 2.5.

**1.3.2.3 Panel Grouping** - Making this selection places the "Panel Grouping" menu in the lower left window pane. This menu allows the user to select panels for grouping into components. The user may change the panels grouped to form an existing component, make a new component or delete an existing component. Panel grouping is discussed in more detail in Section 2.4.

**1.3.2.4 Input Units** - The user can specify the units associated with the input geometry. This information is used to control the units requested for input when defining viscous analyses. This flag *must* be set when performing viscous analyses for proper analysis.

**1.3.2.5 Streamline Input** - Selecting this menu item will launch the streamline tracing program input window. Streamline tracing input is discussed in detail in Section 3.1.

**1.3.2.6 S/HABP Run Setup** - By selecting this menu item the user will launch windows for S/HABP run input. Runs are individual S/HABP analyses such as an inviscid pressure analysis on the body, or a viscous analysis on the wing. Runs are defined individually and then can be executed by any "case" defined by the user. In this manner, the user can specify a run to be executed at a series of Mach numbers (each Mach number being a separate case) without having to enter the same input for each case. Run Setup is discussed in more detail in Section 4.2.

**1.3.2.7 S/HABP Case Setup** - The "S/HABP Case Setup..." menu item launches a window for defining the S/HABP case input. The case input represents all of the variables which remain constant for a given call to the S/HABP aerodynamics program. These variables include flight conditions, reference quantities, and flight attitude ( $\alpha$ - $\beta$  cards). Case input is discussed in more detail in Section 4.1.1.

**1.3.2.8 Trim Input** - Selecting the "Trim Input ..." menu item launches a window so the user can create input to the aerodynamic trim program. The trim program input is described in detail in Section 5.

### 1.3.3 Display

The third menu on the main window menu bar is the "Display" menu which is shown in Figure 1.3-4. This menu allows the user to control what geometry is displayed in the bottom window pane and how it is displayed. Section 2.2 describes in detail all of the options available to the user.

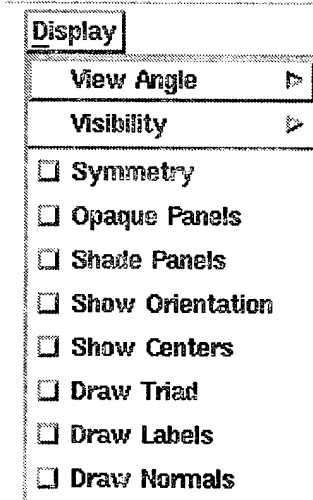


Figure 1.3-4 Display Menu



### 1.3.4 Analysis

The "Analysis" menu executes the selected program using the current input (not the most recently saved input). The analysis menu is shown in Figure 1.3-5.

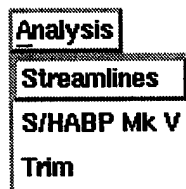


Figure 1.3-5 Analysis Menu

### 1.3.5 Results

The "Results" menu launches the VECC postprocessors. The first two selections shown in Figure 1.3-6 launch the line plotting program thus allowing the user to view and print plots of either trimmed or untrimmed aerodynamic coefficients.

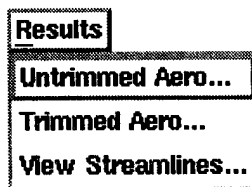


Figure 1.3-6 Results Menu

### 1.3.6 Help

The VECC GUI contains on-line help files which can be accessed through the "Help" menu on the main window or via the help button available on each window. The "Help" menu shown in Figure 1.3-7 allows the user to access help on any of the features of the main window. In addition, a help index is available (the first selection in the "Help" menu) which contains a list of all of the on-line documentation that is available. The individual help windows that are launched are editable Motif text widgets. Thus the user can customize any of the help windows with his/her own comments.

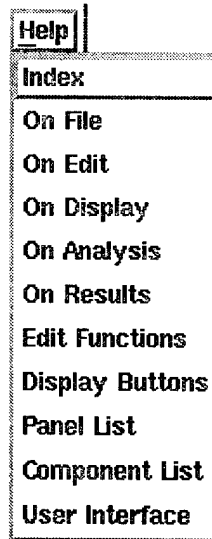


Figure 1.3-7 Help Menu

## 2. GEOMETRY MODELING

The geometry model used by VECC consists of the X, Y, and Z coordinates of points which make up the vehicle external moldline. The vehicle external moldline is subdivided into components, panels, sections, and elements.

A right-handed coordinate system is used. The X axis is aligned with the vehicle longitudinal axis and is positive forward. The Y axis is positive to the left when the vehicle is viewed from behind. The Z axis is positive upward. The panels are usually mirrored (i.e., symmetric) about the X-Z plane such that only the left side (positive Y values) is modeled. However an asymmetric panel can be modeled on either side of the X-Y plane.

The number of elements and grid points used in the model effects both the accuracy and the amount of time required to compute the solution. Extremely detailed modeling of every surface is generally not required and slows down both the drawing and analysis. The amount of detail modeled depends in part on the level of analysis the user is performing. When performing level 1 inviscid pressure analysis for example, flat surfaces can be modeled with only one quadrilateral without affecting prediction accuracy. However when performing level 2 analysis (such as level 2 viscous analysis), the user may need to define all surfaces, even flat surfaces, with more detail. Some geometry modeling guidelines will be given in the Streamline Analysis and S/HABP Analysis sections of this manual.

Many of the inputs and controls are made using the workstation mouse. VECC interprets the mouse movements and button presses in several ways depending on which window is active. The mouse is used to select the following items while using the geometry modeler.

- |               |   |
|---------------|---|
| Left mouse:   | Picking items from menus, activating buttons, and adjusting slide bars.<br>Pick geometry mesh for editing when in 3D-build mode.<br>Activate Display and Edit buttons in the center pane of the main window (making large changes). |
| Middle mouse: | Monitor grid point coordinates of the geometry mesh. Shift clicking allows multiple points to be monitored.<br>Activate Display and Edit buttons (making small changes).  |
| Right mouse:  | Toggle part visibility by clicking on geometry mesh.<br>Activate Display and Edit buttons in the center pane of the main window (select exact changes).   |

## 2.1 Geometry Modeling Conventions

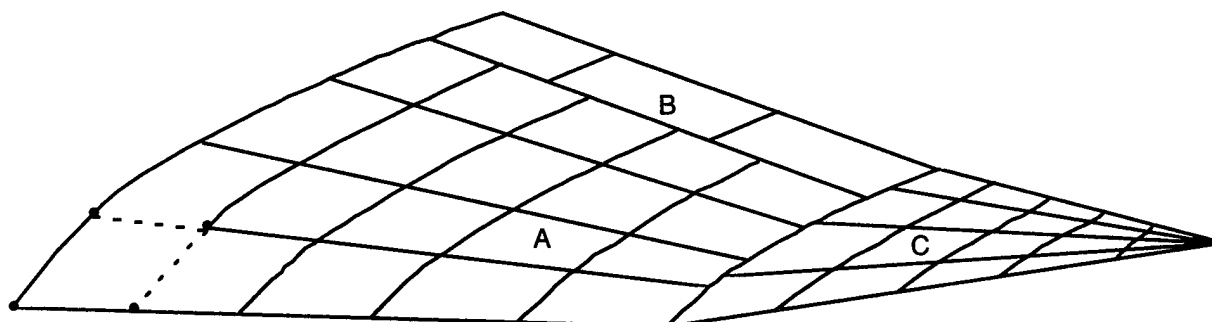
### 2.1.1 Geometry Naming Conventions

Before proceeding with the detailed descriptions of the geometry modeling, several (and often confused) geometry terms should be defined.

Grid Point	A single point defined by its X,Y,Z coordinates.
Surface Element:	This, the smallest geometry unit, consists of four related points on the surface of the vehicle and the area enclosed by lines connecting successive points.
Plane Quadrilateral Element	Each surface element is converted by the program into a plane quadrilateral element. The plane quadrilateral element is the basic geometric unit used in the force calculations. This unit, in effect, is the integration step size and is fixed once the surface element representation of the shape is established.
Cross-Section Cut:	A cross section cut is that view obtained by cutting the vehicle in the longitudinal plane (Y-Z plane), at a constant X-station.
Vehicle Section:	A vehicle <i>section</i> consists of an aggregation of surface elements that have the same number of points per cross section. Panels with more than one section can be modeled in VECC by reading in existing geometries. However, new panels generated in VECC will have only one section per panel.
Vehicle Panel:	A <i>panel</i> consists of one or more sections.
Vehicle Component:	A <i>component</i> is defined as a major part of the vehicle that is to be analyzed by the program as a unit (i.e., a wing, tail, etc.). A component is usually made up of several panels and is the level at which analysis methods are defined.

These various definitions are illustrated in the table below and in Figure 2.1-1.

Basic Quantity	Combine	To Form	Examples
Points	→	Elements	
Elements	→	Sections	Inboard Section A, Flap Section B, Outboard Section C.
Sections	→	Panels	Sections A,B,C form upper wing panel.
Panels	→	Components	Upper and lower panels form wing component.
Components	→	Vehicle	Wing, fuselage, tail, etc. components form the complete vehicle.



**Figure 2.1-1, Geometry Definitions**

Each geometry panel and component is named with up to 20 characters for descriptive purposes. The names are shown in the center pane of the main modeler window. A panel or component name can be changed by clicking on it with the mouse and typing in the new name.

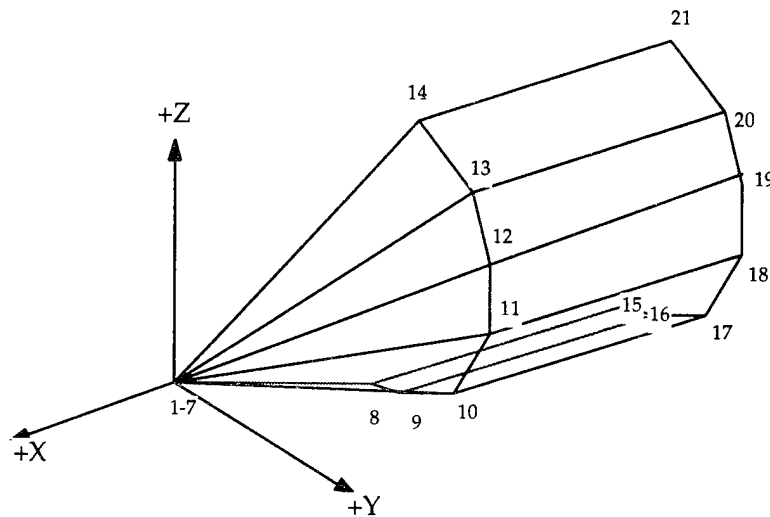
The analytical methods in S/HABP treat each component on an individual basis so that the points of neighboring parts need not be in perfect alignment.

### **2.1.2 Grid Point Orientation**

Grid points must be properly ordered to insure the surface normal lies in the outward direction. One way to determine the correct point order is to imagine placing the palm of your **left** hand on the surface. Fingers should point in the direction of ascending points in the cross section. Your thumb will then point in the direction each subsequent cross section should be built. For

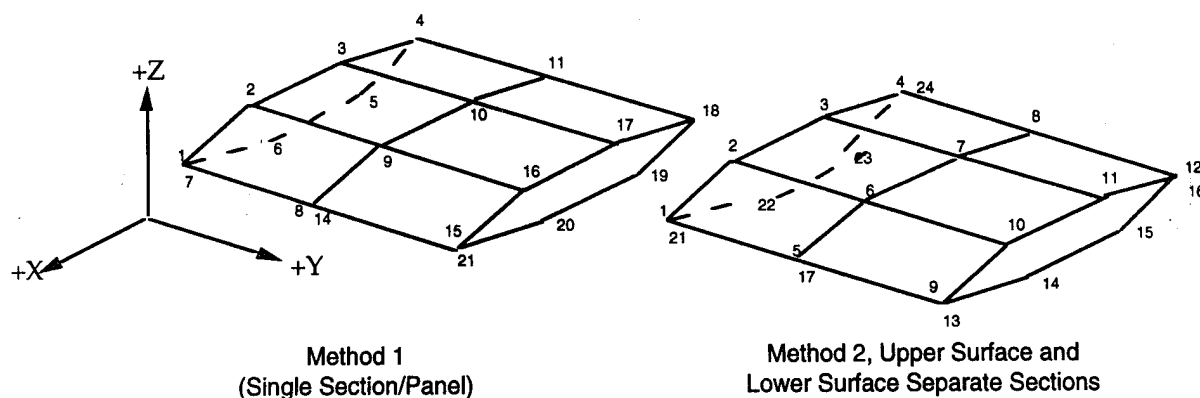
example, on a body like the one shown in Figure 2.1-2, placing your **left** palm on the body with the heel of your hand on the bottom centerline and fingers pointing upwards toward the top centerline causes your left thumb to point aft (negative X-direction). Therefore, the cross sections start at  $x=0$  and progress in the negative X-direction. Figures 2.1-2, 2.1-3, and 2.1-4 show the correct ordering of points and cross sections.

Grid points on bodies generally start at the bottom centerline of the cross section and proceed in a counterclockwise direction, when viewed from the front of the component. With each subsequent cross section moving down the negative X-axis.

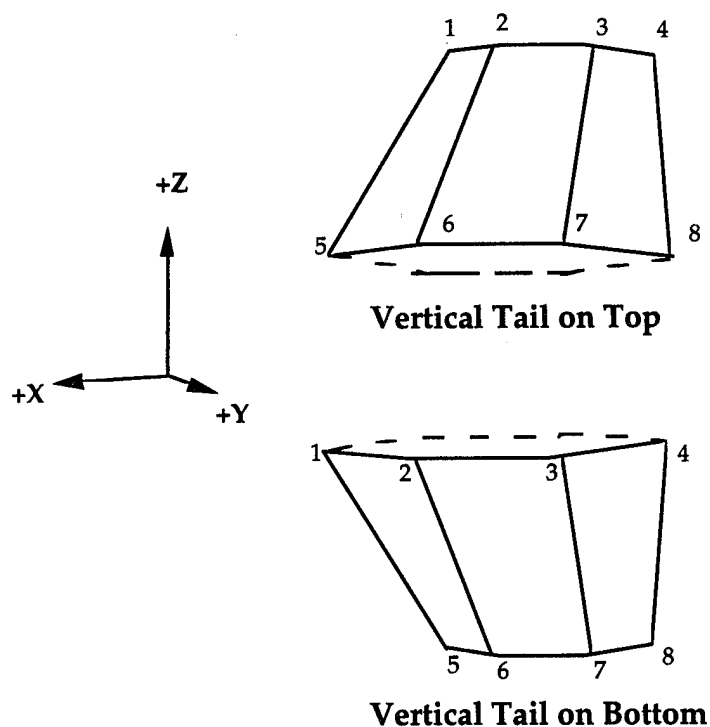


**Figure 2.1-2 Point Numbering for Body**

The cross sections of the fins, wings or tails (generically referred to here as aerosurfaces) are usually defined in the streamwise direction (the aerosurface generator creates cross sections in the streamwise direction at constant Y values) beginning at the root chord. The cross sections are ordered from root to tip, and the points are ordered from leading edge over the upper surface to the trailing edge, and around the lower surface to the leading edge; thus the leading edge point is both the first and last points listed for each cross section as shown in Figure 2.1-2. When modeling vertical tails, only the part which lies in the positive Y-direction is defined as shown in Figure 2.1-4. This is the typical manner for representing a wing, however aerosurfaces are not required to be defined in this manner. For example, the upper surface may be a separate panel from the lower surface, and the leading edge may be separate from both the upper and lower surface. It is only important to keep in mind the direction of the surface normal to ensure that it is facing outward. The VECC modeler can draw the surface normals by selecting "Draw Normals" from the "Display" menu.



**Figure 2.1-3 Point Numbering for Aerosurfaces**



**Figure 2.1-4 Point Numbering for Vertical Tails**

### 2.1.3 Maximum Model Sizes

The maximum number of points on a geometry model is limited by internal array sizes. The array sizes can be changed by modifying header files and recompiling the source code. The program is normally compiled with the following limits:

- maximum number of components = 15
- maximum number of panels = 25

- maximum number of sections = 35
- maximum number of panels per component = 10
- maximum number of sections per panel = 10
- maximum number of cross sections per panel = 20
- maximum number of points per cross section = 40
- maximum number of elements = 2000
- maximum number of points = 2000

In order to limit the array sizes (and thus memory requirements), not every parameter can be maximized at the same time. For example, the maximum number of panels is 25 and the maximum number of sections is 35. However, a panel can consist of up to 25 sections. Warning messages will appear if these maximums are exceeded.

## 2.2 Model Display Modes

The user has several ways to control the viewing of the geometry model when building or verifying the geometry. This section discusses the "Display" menu and the display controls of the center window pane.

### 2.2.1 View Control

View controls are used to examine the vehicle model from various aspects. The view orientation is controlled by using the display buttons shown in Figure 2.2-1 (which are in the center pane of the main window), the model scroll bars, and the Display>View angle menu. Changing the view has no effect on the configuration inputs.

The Display>View angle menu allows selecting from several standard orientations such as front, top, or side. Selecting a standard display angle also resets the pan and zoom factors so that the entire vehicle fits within the window.

Display	
	Zoom
X-Center	Roll
Y-Center	Pitch
Z-Center	Yaw

Figure 2.2-1 View Control Buttons

To use the view control buttons, click and hold the left mouse button over the desired function, then drag the mouse left or right. Clicking with the middle mouse button (instead of the left



mouse button) allows finer changes to be made. Clicking with the right mouse button displays a menu of discrete values which can be picked to make exact changes. These selection options apply to each of the view control buttons shown Figure 2.2-1.

The roll, pitch, yaw, and zoom buttons control the angle and size of the vehicle with respect to the screen. These controls are applied with respect to the center of rotation which is the center of the geometry display window pane.

The X, Y, and Z center buttons are used to change the center point which is used for rotation and zoom control. By default, the geometric center of the vehicle is located at center of the geometry display window pane. Thus the center of the geometry is the center of rotation for view control. Changing the center of rotation is useful when examining other regions such as the nose or tail.

The slide bars along the bottom and right edges of the display window are used to pan the image left/right or up/down. Panning shifts the image only and does not alter the center of rotation as does the view center controls.

### 2.2.2 Invisibility

Complex geometry models can be difficult to edit due to the large number of lines and grid points displayed on the screen. Using the invisibility control allows you to reduce the number of lines drawn, making it easier to select specific points or sections. When a part of the geometry is invisible, it cannot be selected, however it is still part of the configuration and is included in all analysis functions. Making regions invisible also increases the drawing speed. The "Invisibility" submenu is shown in Figure 2.2-2.

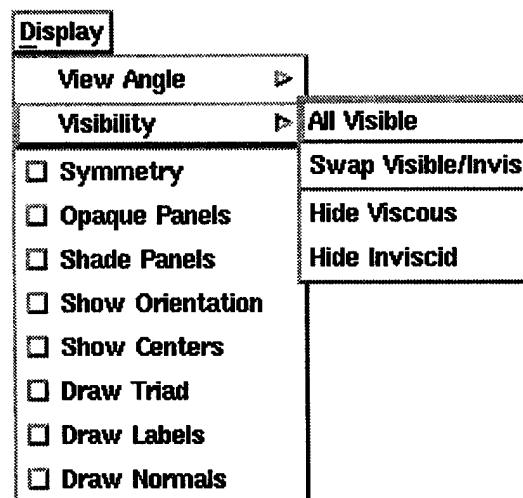


Figure 2.2-2 Invisiblity Menu Options

Upon entering the GUI all of the panels are visible. Geometry can be made invisible on a per panel or per component basis. To make a geometry panel invisible, click on the wire mesh with the right mouse button. Panel and component visibility can also be controlled by the button next to their name in the "Panels" and "Components" dialog area. If the button is depressed, the panel or component is visible. Clicking a component button (making component either visible or invisible) causes the corresponding panel buttons to be in the same position. This can be useful in determining which panels comprise a component and where they are located. For example if you are working with a geometry model someone else developed, make all of the components invisible then toggle them on one at a time. Each panel of the component is displayed and the button next to the panel name(s) is also depressed.

After making geometry invisible, it can be redisplayed by selecting Display>Invisible>All visible.

The Display>Invisible>Swap Visible/Invis function reverses the current visibility state of every panel or component. This is a useful shortcut for making only a small amount of the total geometry visible. To accomplish this, first make the panels which you would like to view invisible, then use the swap function to erase the remainder of the model and view the selected panels.

The final option for the geometry visibility is to "Hide Viscous" or "Hide Inviscid" geometry. A separate "viscous" geometry with fewer elements is often defined when doing a level 1 viscous analysis. To help organize input, panels can be assigned the characteristic of being either viscous or inviscid (see Section 2.5). The "Hide Viscous" or "Hide Inviscid" options under Display>Invisibility allow all panels of a given type to be made invisible. The user is advised to make all panels visible before selecting this option to avoid confusion.

### **2.2.3 Display Options**

The GUI main window menu bar provides several options which control the vehicle display window. Display options are used to enhance the clarity of the vehicle and have no effect on the analysis. The "Display" menu is shown in Figure 2.2-3.

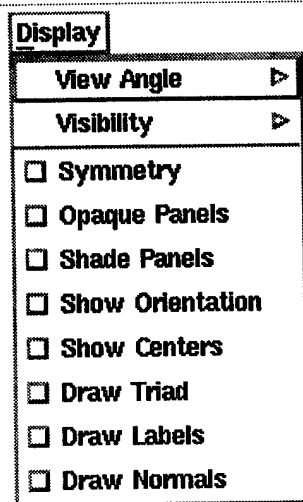


Figure 2.2-3 Display Menu Options

**2.2.3.1 Symmetry** - The default display is to draw only one half of the vehicle. However when symmetry is turned on, both sides of the geometry are drawn (for panels that are symmetric, see Section 2.5). This can be useful when printing the geometry.

**2.2.3.2 Opaque Panels** - This option causes the geometry model to be displayed with hidden line removal. The painters algorithm is used to remove hidden lines. That is, the element is drawn as a filled polygon and the element whose centroid is furthest from the users point of view is drawn first. This algorithm can lead to incorrect hidden line removal if elements become too large. For example, a constant cross section body is often defined by long single elements. If the body has tail fins, their element centroids are located aft of the body element centroids, and the fin elements would be drawn behind the body elements. To improve the display, the user can add a cross section just forward of the fins. Turning panels off decreases the drawing time resulting in a smoother display.

**2.2.3.3 Shade Panels** - Shading panels causes a shaded polygon to be drawn for each quadrilateral element in the geometry model. Each panel is shaded in a different color. The hidden line removal is accomplished using the same algorithm as the opaque panels option.

**Note:** In shaded panel mode, the hidden line removal algorithm may cause a delay before a correct image is rendered. The model will be drawn twice. The first time all the panels will be drawn in color. The second pass will incorporate the hidden line removal algorithm results.

**2.2.3.4 Show Orientation** - This option allows the cross section orientation to be displayed. After selecting "Show Orientation," the cross sections are drawn in black while the lines

connecting the cross section are drawn in blue. It is necessary to define the panel orientation both for S/HABP and for streamline tracing (see Section 2.5). This option can be very helpful when working with old geometry files or geometry models developed by others.

**2.2.3.5 Show Centers** - By selecting "Show Centers," the element centroid is drawn as a red dot. The element number can be displayed by clicking on the element centroid with the middle mouse button. Use shift/middle mouse to list multiple elements. Note that this brings up a panel number and an element. This pair of integers uniquely defines the element on the geometry. The user will find this option helpful when doing streamline tracing and for defining level 1 viscous input.

**2.2.3.6 Draw Triad** - The triad is used to indicate the X, Y, Z axis orientation by displaying a short line segment in each direction. The triad is located at the vehicles geometry system origin.

**2.2.3.7 Draw Labels** - Labels are used to identify each geometry panel. When labels are turned on, the name of each panel is shown with a leader line drawn to it. The position and orientation of the labels are determined by the current viewing angle and cannot be changed. Figure 2.2-4 shows a display with labels turned on.

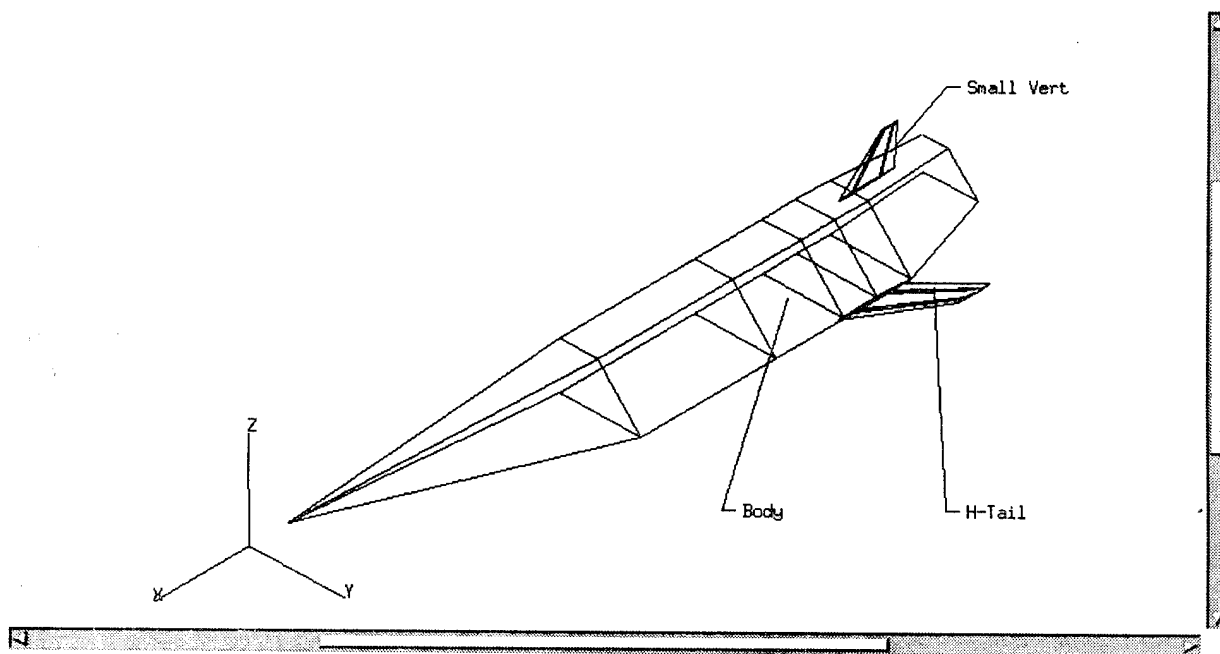


Figure 2.2-4 Modeler Display with Triad and Labels

**2.2.3.8 Draw Normals** - The "Draw Normals" option draws arrows at the element centroids in the direction of the surface normal. This is useful to determine if panel grid points have been input in the proper order.

#### 2.2.4 Grid Point Monitor

The grid point monitor allows you to constantly view the X, Y, Z coordinates of individual geometry grid points. Monitoring points is useful when aligning parts or to accurately position points or cross sections.

The coordinates of monitored points are shown in the lower right portion of the display. The coordinates are constantly updated as the geometry is being edited. Points which are monitored appear with a small blue circle around them in the geometry display.

Use the middle mouse button to select grid points which you would like to monitor. Multiple points may be selected by holding down the shift key while picking. Up to 20 points may be monitored at a time. To cancel grid point monitoring, press the middle mouse button with the cursor anywhere in the background.

#### 2.3 3D Geometry Build

The VECC geometry modeler allows the analyst to quickly create panels or make modifications to existing geometries. It includes options which allow the user to move, scale, roll, pitch and yaw any panel of the model. It can also be used to access the geometry library which contains common shapes which may be used to construct a model from scratch. Editing functions can be performed at the panel, cross section or grid point level.

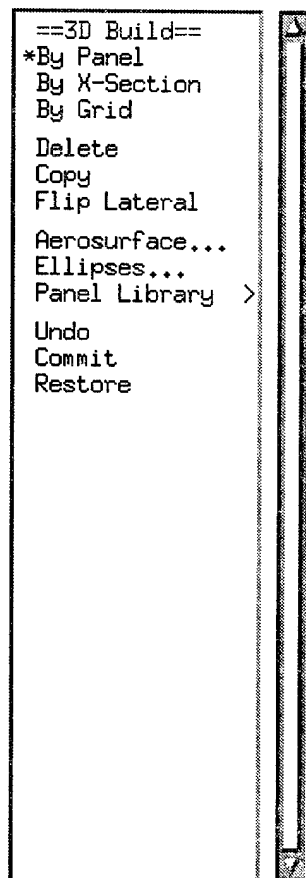
**Hint:** When shaping a new vehicle geometry, keep in mind that a model with fewer cross sections and points is easier to edit. After the approximate shape is obtained, more sections and points can be added using the copy function. Continue increasing the mesh density until the desired level of detail is obtained. Similarly, to change the shape of existing models, it is often easier to first delete some of the points and cross sections before making any modifications to the geometry.

The geometry modeler displays menus along the left side of the display window which are used to access the geometry and panel grouping functions. The functions available in the menu depend on the current editing mode. The editing mode can be either 3D Build or Panel Grouping and is selected under the "Edit" menu on the menu bar.

Select from the modeling menus with the left mouse button. Menus which have a ">" on the right side have additional options. These submenus are presented in a marching menu format (i.e., new options are added below the existing menu). Menu selections which are grayed out can not be selected. When there are more menu items than will fit in the window, a slide bar

appears allowing the menu to be shifted vertically. The 3D Editing menu is shown in Figure 2.3-1.

Menu selections which are followed by several dots "..." indicate additional keyboard input will be required to complete the command. When the keyboard window appears, type the necessary additional text and press enter. Pressing the escape key cancels the command.



**Figure 2.3-1 3D Build Editing Menus**

Geometry building is most often performed one panel at a time. Large changes to the panel are made by first selecting the entire panel and adjusting its overall size and shape. Further changes are then made by selecting individual cross sections and forming the desired cross section shapes. More detailed shaping can be completed by selecting grid points and moving them individually.

### **2.3.1 Selecting Panels, Cross Sections or Grids**

Geometry regions are selected by placing the cursor over the wireframe mesh and pressing the left mouse button. The area selected is then drawn highlighted. Shift click a highlighted region

a second time deselects it. Selected geometry can then be manipulated using the editing functions.

Multiple regions can be selected by holding down the shift key (shift click). The entire configuration can be picked by holding down the alt key while clicking.

**Note:** Geometry which has been set invisible cannot be selected. To simplify picking on a complex vehicle, make unwanted geometry invisible.

### 2.3.2 Geometry Editing Buttons

The geometry editing buttons are located in the center pane of the modeler window. Before using an editing function, you must select a geometry region as described in Section 2.3.1. When a function is activated, the change is applied to the currently selected region. To apply an editing function, click with either the left or middle mouse button (coarse or fine editing respectively) and, while holding the button down, drag the mouse to the left (decrement) or right (increment). A status of the edit is shown below the edit functions (for example the exact scaling that would be performed if the mouse button were released). Clicking an editing function with the right mouse button displays a series of discrete increments which may then be selected with the left mouse button. The editing buttons are shown in Figure 2.3-2 and their functions are described below:

#### Edit Functions

<b>Scale</b>		
<b>X-Scale</b>	<b>X-Trans</b>	<b>Roll</b>
<b>Y-Scale</b>	<b>Y-Trans</b>	<b>Pitch</b>
<b>Z-Scale</b>	<b>Z-Trans</b>	<b>Sweep</b>

Figure 2.3-2 Geometry Editing Buttons

<b>Scale</b>	The selected region is scaled equally in the X, Y, and Z directions. Each panel is scaled about the first point on its first cross section. Panels and cross sections may be scaled but scaling of individual grid points is ignored.
<b>X Scale</b>	The selected region is scaled in the longitudinal direction.
<b>Y Scale</b>	The selected region is scaled in the lateral direction.
<b>Z Scale</b>	The selected region is scaled in height.
<b>X Trans</b>	The selected region is moved forward or aft. When moving cross sections care must be exercised not to change the order of cross sections by moving past adjacent sections.
<b>Y Trans</b>	The selected region is moved sideways.
<b>Z Trans</b>	The selected region is moved up or down.

<b>Roll</b>	The roll orientation of the selected panel is changed. The panel is rotated about the X-axis with the first point of the panel as the origin of the axis of rotation.
<b>Pitch</b>	The panel is rotated about the Y-axis with the first point of the panel as the origin of the axis of rotation.
<b>Yaw</b>	The panel is rotated about the Z-axis with the first point of the panel as the origin of the axis of rotation.

**Hint:** The sensitivity of the translate functions is proportional to the current zoom factor. To make finer changes to the model, enlarge the picture with the zoom display button. To make very coarse large changes, shrink the picture.

### 2.3.3 Geometry Editing Menus

When in the 3D geometry edit mode, the following menu items appear on the left side of the display window.

**2.3.3.1 By Panel** - Allows entire panels to be selected by placing the cursor on the mesh and pressing the left mouse button. Selected panels will be highlighted in red. When "By Panel" is pressed, any existing selections are canceled.

**2.3.3.2 By Cross Section** - Allows individual cross sections to be selected by placing the cursor on the cross section mesh and pressing the left mouse button. Selected cross sections will be highlighted in red. When "By Section" is pressed, any existing selections are canceled.

**2.3.3.3 By Grid** - Allows individual grid points to be selected by placing the cursor on the grid point and pressing the left mouse button. Selected grid points will appear with a red circle drawn around them. When "By Grid" is pressed, any existing selections are canceled.

**2.3.3.4 Delete** - The selected panels, cross sections, or points are deleted. When deleting points, note that the corresponding point of every cross section is automatically deleted. This must be done to maintain a constant number of points per cross section.

**2.3.3.5 Copy** - The selected panels, cross sections, or points are copied. When a panel is copied, it is duplicated just aft of its original position. Cross sections are copied by adding a section midway between the *selected* section and the next one. The new cross section shape is interpolated from those on either side of it. Points are copied by adding a grid point midway between the selected grid and the next one. When copying points, note that a grid point is added to every cross section on that part. This is done to insure that a constant number of points per cross section exists.



If the shift key is held down when a cross section is copied, the new cross section is added midway between the selected section and the next one, however its shape is identical to the selected section.

**2.3.3.6 Flip Lateral** - The selected panels are moved to other side of geometry.

**2.3.3.7 Aerosurface** - Opens the aerosurface (wings, tail, etc.) generator window which is discussed further in Section 2.3.4.

**2.3.3.8 Ellipse** - Opens the ellipse (body, nacelle, etc.) generator window which is discussed in detail in Section 2.3.5.

**2.3.3.9 Panel Library** - Extract a panel from the Panel Library. The panel library is opened via the "Geometry Database ..." menu option from the "File" menu.

**2.3.3.10 Undo** - The last geometry editing operation is reversed and any selections are canceled. Only the *very last* function can be undone.

**2.3.3.11 Commit/Restore Geometry** - The restore command returns the geometry model to the shape saved by the last commit command. Using commit often when making changes to the geometry allows easy recovery from errors due to incorrect inputs.

Using commit does *not* write the geometry out to a file. Use *File>Save* to save the configuration on the disk.

## **2.3.4 Aerosurface Generator**

An aerosurface generator has been incorporated into the geometry modeler that effectively takes the place of the wings, fins, canards/horizontal tail geometry routines which are contained within S/HABP. The aerosurface generator window is launched by clicking on the "Aerosurface ..." button of the "3D Build" menu (see Section 2.3.3). This window allows the user to define an aerosurface (wing, fin, tail, etc.) in one window. Only the root and tip chord are generated each of which can have a separate airfoil section. Additional cross sections can be added by copying the root chord and then modifying it using the edit functions in the middle window pane. The "Aerosurface Generator" window is shown in Figure 2.3-3.

	Root	Tip
Chord Length:	20.0	10.0
X-station, LE:	-50.0	-60.0
Y-station, LE:	5.0	20.0
Z-station, LE:	0.0	0.0

**Airfoil Shapes**

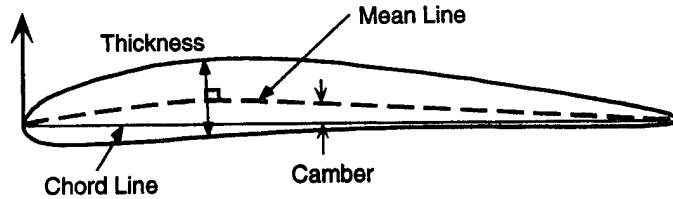
  

% Chord Location	Root		Tip	
	% Thickness	Camber	% Thickness	Camber
0.00	0.00	00	0.00	00
30.00	8.00	00	8.00	00
70.00	8.00	00	8.00	00
100.00	0.00	00	0.00	00

**Figure 2.3-3 Aerosurface Generator Window**

The user supplies the root and tip chord lengths, XYZ coordinates of the leading edge point, and the thickness/camber distribution. The chord length and XYZ leading edge stations are input in the geometry coordinate system (units and origin). The airfoil sections are defined as a percentage of the chord length. By allowing the user to input the leading edge point, he/she has control over the sweep of the aerosurface. Twist and dihedral can be added with the 3D Build Edit Functions after the aerosurface has been made. Once input has been completed click "OK" to generate the new aerosurface panel and return to the GUI main window or click "Apply" to add the new panel and begin input on another aerosurface. NOTE: Do not click both "Apply" and "OK" if you intend to generate only one aerosurface because each action would generate a new panel.

The input column entitled %Chord Location ranges from 0 to 100 (positive numbers). The %Thickness column is the thickness of the uncambered airfoil (i.e., normal to the mean camber line) as a percentage of the chord length. The column labeled "Camber" is the height of the mean line above the chord line as a fraction the chord length. These terms are illustrated in Figure 2.3-4.



**Figure 2.3-4 Input Definition For Airfoil Shapes**

Available under the pop-up menu labeled "Airfoil Shapes" are some predefined airfoil sections. Click on the menu to open it. Use the **right** mouse button to select the desired airfoil shape. After an airfoil shape is selected, its definition fills both the root and tip chord definition. The predefined airfoil shape can then be customized if desired.

Upon return to the main window, the new aerosurface is placed horizontal to the X-Y plane at the user specified coordinates. The aerosurface can then be rotated, translated, twisted, etc. to fit the user's needs. If the aerosurface is to be a vertical tail lying on the centerline, the leading edge point (which is repeated) and all points with -Y coordinates should be deleted or the panel should be labeled as asymmetric (see Section 2.5).

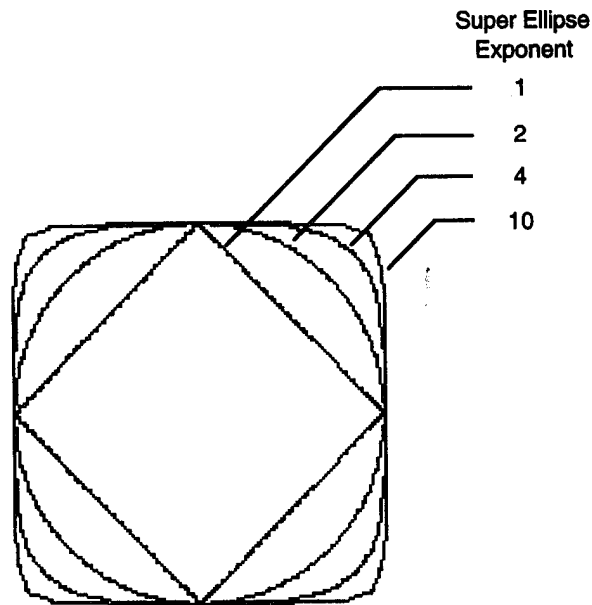
### **2.3.5 Ellipse Builder**

The ellipse builder allows the user to create cylindrical panels with or without noses and boattails. The ellipse builder window shown in Figure 2.3-5 is launched by selecting "Ellipses ..." from the 3D Build window.

Ellipse Builder	
Total Length:	80.0
Start Angle:	0.0
End Angle:	180.0
No. of Points:	14
Nose Shape:	None <input type="checkbox"/>
Nose Exponent:	2
Nose Length:	10.0
Nose Width:	10.0
Nose Height:	10.0
Boattail Shape:	None <input type="checkbox"/>
Boattail Exponent:	2
Boattail Length:	10.0
Boattail Width:	10.0
Boattail Height:	10.0
Base Area:	0.0
<input type="button" value="OK"/> <input type="button" value="Apply"/> <input type="button" value="Cancel"/> <input type="button" value="Help"/>	

Figure 2.3-5 Ellipse Builder Window

The cylindrical panels can be elliptical or "super ellipses" in cross section. The shape of a super ellipse is controlled via an exponent which determines how "square" each corner is as shown in Figure 2.3-6. An exponent of 2 generates ellipses, and as the exponent increases, rounded corners develop. All of the cross sections shown have equal width and height. However the user can specify any width/height ratio.

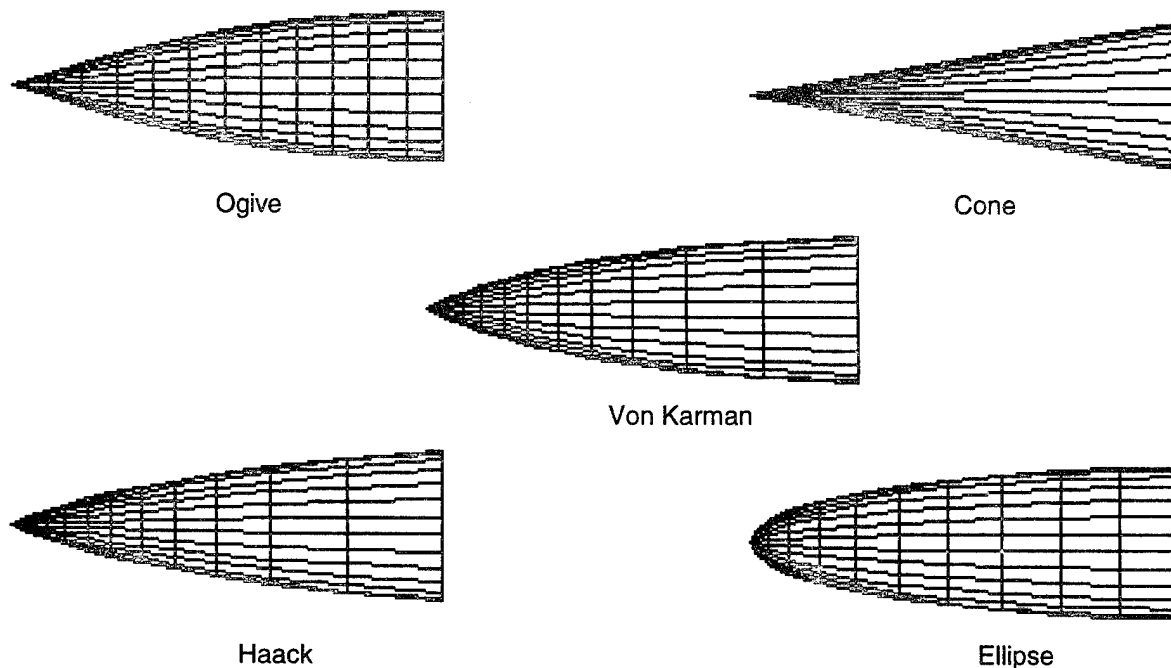


**Figure 2.3-6 Sample Cross Sections Modeled With The Ellipse Builder**

The user can create a complete cross section (i.e., from bottom centerline to top centerline) or only a portion of an ellipse. The circumferential extent of the cross section is controlled via the Start Angle ( $0^\circ$  for bottom centerline) and End Angle ( $180^\circ$  for top centerline). Two cross sections will be built. The first cross section is determined by the "Nose Exponent," "Nose Width," and "Nose Height" even if a nose is not desired. The base cross section is controlled via the "Boattail Exponent," "Boattail Width," and "Boattail Height" input parameters even if a boattail shape of "none" is selected.

A nose may be modeled by choosing a nose shape from the "Nose Shape" option menu. Nose shapes are formed by a series of 12 cross sections which have a uniform curvature when viewed from the top or side. The user specifies the nose length, nose width, nose height, and nose exponent (controls cross section shape as discussed earlier) which define the cross sectional shape at the base of the nose. The input nose width and height are for the width and height of the geometry including the portion reflected in the negative X-Y plane (e.g., it is the diameter for circular cross sections). Available nose shapes, listed below, are shown in Figure 2.3-7.

Ogive	Sharp Tangent Ogive
Cone	Sharp Cone
Von Karman	L-D Haack (length-diameter constrained)
Haack	L-V Haack (length-volume constrained)
Ellipse	Profile of $\frac{1}{2}$ ellipse



**Figure 2.3-7 Nose Shape Options**

Boattail shapes may also be defined in the ellipse builder using essentially the same input as necessary for generating a nose shape. When the user specifies the boattail width, boattail height and boattail exponent, he/she is defining the cross section at the beginning of the boattail. The base of the panel is defined via the base area input. The cross sectional shape is then held constant while the height and width are scaled to form the requested boattail shape (ogive for example).

### **2.3.6 Geometry Part Library**

The geometry library contains commonly used panels which can be easily added to a configuration. The library menu shows a list of each panel in the geometry database. When a panel is selected from the library menu, its geometry is copied from the database into the current configuration.

A new vehicle can readily be created from the part library. To do so, start the GUI with a new filename and then use the part library function on the editing menu to select the desired body, wing, and fin parts from the geometry database. Delete any cross sections and grid points that are not critical to the geometry definition and shape as needed using the editing functions.

To switch to another geometry library, select "*File>Geometry Database...*". The default database name is specified in the application resource file.

To add new panels to the geometry library, first select the desired panels with the mouse, then select "*File>Add to Database*". Panels can be deleted from the database using a text editor to remove the lines associated with the unwanted panel from the database file.

**Note:** You must have write privilege to the geometry database to add panels to it. File permissions are under control of the system administrator.

## **2.4 Panel Grouping**

Panels are grouped to form components for S/HABP analysis. The VECC GUI provides the user with a graphical means of defining components. To create components, select "Panel Grouping" from the Edit menu of the main window. This brings up the "Grouping" menu on the lower left window pane. The user should select "Add Component" on the "Grouping" menu. A new unnamed component will appear in both the lower left window pane and in the "Components" dialog of the middle window pane. The panels which will form this component are then selected by clicking on them, and they will appear highlighted in red. A panel can be "deselected" by clicking on the panel again.

The user may change which panels are grouped by clicking on the name of the component to be changed (note: an asterisk will appear next to the active component name) in the "Grouping" menu. Panels are either added to the component or taken from it by clicking on the panel in the geometry display window in the same manner as they were originally selected. The panels which appear highlighted in red are those panels which belong to the active component (denoted with an asterisk). The same panel can be included in more than one component.

Component names are changed in the "Components" dialog area (middle window pane) by clicking on the name, deleting the existing name, and typing the new name. Note that the name is not updated under the "Grouping" menu until another action is taken in that menu (i.e., any button in the "Grouping" menu is clicked).

**Note:** Components must be defined before the user can begin input of S/HABP runs.

## **2.5 Panel Type**

The "Panel Type" window shown in Figure 2.5-1 is launched from the "Edit" menu. In this window, the user specifies attributes of each of the panels. These attributes include cross section orientation, panel symmetry, and whether the panel is "inviscid" or "viscous."

The cross section orientation is the same as the variable IORN in S/HABP Mark IV. "X-sect" indicates that the points defining the cross section are at a constant (or nearly constant) x-position and thus define the panel cross section. "Stream" indicates that the points of the panel are defined in the streamwise direction which is common for wing definitions. The orientation of a panel can be changed by clicking on the words "X-sect" or "Stream."

The panel symmetry characteristics are defined by the "Symmetric" button. If the button is pushed in, the panel is assumed symmetric about the X-Z plane and only the left side of the geometry is input. S/HABP will mirror the panel to define the right side. If the button is out, the panel is not symmetric. Thus S/HABP will analyze only the input geometry and will not account for a mirrored panel.

Designating a panel as "Viscous," "Inviscid" or both is a new concept employed in the VECC GUI. This additional panel characteristic is used to organize the S/HABP input windows and controls which panels are available for analysis. To designate a panel as either viscous or inviscid, push in the button below the appropriate label. The same panel can be used for both inviscid and viscous analysis by pushing both the viscous and inviscid button for that panel. By default, new panels are inviscid panels.

Orientation	Symmetric	Viscous	Inviscid	Panel Name
X-sect	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	Body
X-sect	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	Small Vert
X-sect	<input type="checkbox"/>	<input type="checkbox"/>	<input type="checkbox"/>	H-Tail

Figure 2.5-1 Panel Type Window

## 2.6 Geometry Units

The user should specify the units of the geometry via the "Input Units>" submenu of the main window "Edit" menu which is shown in Figure 2.6-1. Specifying the input unit is necessary when performing either level 1 or level 2 viscous analysis. This flag should be set before entering the viscous input windows since it changes the units expected for the viscous input windows. The level 1 S/HABP method has not been changed, and the input for this analysis is converted by the GUI from the user specified units to feet before writing the S/HABP input file. This submenu is a good example of "radio" buttons which are displayed as diamonds. Recall that radio buttons are used whenever only one option can be selected since pushing one of them causes the others to pop out.



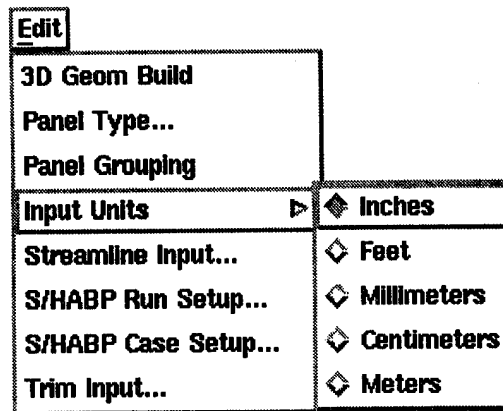


Figure 2.6-1 Input Units Submenu

### 3. STREAMLINE ANALYSIS (QUADSTREAM)

This section discusses the inputs required to run QUADSTREAM and the applicability of the code for generating streamlines. Included in this section is a discussion on program limitations and error messages. An example case is provided to illustrate the use of the program options and examples of the output files are included for this case.

QUADSTREAM is capable of tracing streamlines on complex configurations with multiple components described by quadrilateral elements. QUADSTREAM is based on the Newtonian steepest descent method, which uses only the element inclination angle relative to the velocity vector to determine the streamline trace. The only information required to generate streamlines using QUADSTREAM is a quadrilateral description of the geometry and the flight attitude of the vehicle.

The first step in generating streamlines is defining the geometry which is done in the geometry modeler. After defining the geometry but before entering the streamline program, the user must define the orientation of each panel and group panels into components. Upon entering the QUADSTREAM program, the geometry defined by the geometry modeler is converted into QUADSTREAM format which includes eliminating zero area elements and reordering node IDs. Once the geometry is in the correct format, the user must group the S/HABP defined components into QUADSTREAM parts. These parts are used to start and stop groups of streamlines. Next the user must enter the vehicle attitude, program control variables, and streamline starting/stopping positions. After all the information is entered, QUADSTREAM calculates the streamline starting positions including the element ID for each streamline requested. QUADSTREAM then calculates the x,y,z coordinates of each streamline trace and writes them to an output file. Warning messages are given if a streamline terminates prematurely to inform the user of the specific problem. These warning/error messages are discussed further in Section 3.5, Limitations.

#### 3.1 QUADSTREAM Inputs

Before entering the streamline generation program, the user must first define a geometry in the geometry modeler. After defining the geometry the user must then define the orientation of each panel and group panels into components (See Section 2 Geometry Modeling). The user should keep in mind that QUADSTREAM will use these components to start and stop sets of streamlines. Once the geometry is defined and the panel information entered, the user can edit the streamline inputs required to trace streamlines on the defined geometry. The QUADSTREAM input window is shown in Figure 3.1-1.

**Part Definition:**

Quadstream Part: ☐ Body

**Component List:**

body  
tail  
fin

**Flight Conditions:**

Alpha, Beta (deg)  
0.000 , 0.000

Geometry Gap Tolerance: 0.0100

Geometry Symmetry: ☐ Half Geometry

Streamline Direction: ☐ Front-to-Back

Additional Output: ☐ None

Streamline Definition: ☐ Two Points per Element

Number of Points per Streamline:

Terminate Streamlines That Encounter the Plane of Symmetry: ☐ No

[Edit Streamline Starting/Stopping Conditions...](#)

**Message Box:**

**\*\*WARNING\*\*** Unable to determine group of elements at X = -212.373  
**\*\*ERROR\*\*** Distributed Starting Option failed !  
**\*\*ERROR\*\*** Cannot distribute streamlines at T.E. when tracing FRONT to BACK !

**Figure 3.1-1 QUADSTREAM Input Window.**

The streamline inputs are registered within the program when the user clicks on either the APPLY button or the OK button on the QUADSTREAM window. As part of the process of recording the streamline data, the inputs are checked for validity. If any of the inputs are invalid, a warning message pertaining to the specific problem is written into the message dialog on the QUADSTREAM input window and an error dialog appears informing the user that the inputs are invalid. At this point, the user can change the invalid inputs or click on the CANCEL button to exit the window. The user should be warned that clicking on the CANCEL button will cause all input data to be lost since the last time either the APPLY button or the OK button was pressed.

Once the streamline inputs have been entered and recorded, the QUADSTREAM program can be executed using the Analysis pull down menu on the VECC main window. During the streamline analysis, any warning message is written into the status dialog on the VECC main window. See Section 1.3.4 for more information about executing the QUADSTREAM program.

Access to the QUADSTREAM program inputs is through the Edit Pull down Menu on the Main Window as described in Section 1.3.2. The user simply clicks on the "Streamline Input..." menu

item button which launches the QUADSTREAM input window. The QUADSTREAM window is the main streamline input window and provides access to all streamline inputs. The window is divided into four main areas: part definition, flight conditions, program control flags and streamline starting/stopping conditions. Each area will be discussed in greater detail in the following sections.

### 3.1.1 Part Definition

The part definition dialog area as shown in Figure 3.1-2, allows the user to group S/HABP components into QUADSTREAM parts. The dialog area is separated into two areas, a list of QUADSTREAM parts, and a list of components defined in the geometry modeler. To group components into QUADSTREAM parts, first, select the desired part from the list of parts and then select the desired components to be included in the part definition. If a previously grouped component is selected for inclusion in another part, the component is removed from the first part and included with the current part. All components that are not selected are ignored by QUADSTREAM. Also, only those QUADSTREAM parts that have components associated with them are included in the streamline analysis.

<b>Part Definition:</b>	Body
<b>Quadstream Part:</b>	Wing <input checked="" type="checkbox"/>
	Tail
<b>Component List:</b>	Fin1
fwd	Fin2
aft	Flap

Figure 3.1-2 Part Definition Inputs

### 3.1.2 Flight Condition

The flight condition dialog area as shown in Figure 3.1-3, allows the user to enter up to 10 angle-of-attack/sideslip angle combinations. Each angle-of-attack/sideslip angle combination is entered on a separate line. The angle-of-attack is entered first, followed by the sideslip angle separated by a space or comma. Additional angle-of-attack/sideslip angle combinations can be entered anywhere in the list, one combination per line. Angle-of-attack/sideslip angle combinations can be deleted by deleting the values on a single line (Note: The blank line will remain until the OK button is clicked.). Modifications can be made to an angle-of-attack/sideslip angle combination by moving the pointer to the line displaying the combination desired and editing the values.

**Flight Conditions:**

Alpha, Beta (deg)

0.000 , 0.000	<input type="button" value="Up"/> <input type="button" value="Down"/>
5.000 , 0.000	
10.000 , 0.000	

**Figure 3.1-3 Flight Conditions Input**

### 3.1.3 Program Control Variables

The program control variables are used to control various aspects of the QUADSTREAM program. All inputs have default values which should suffice for most cases.

**3.1.3.1 Gap Tolerance** - The geometry gap tolerance input is shown in Figure 3.1-4 and is used to determine which elements lie on adjacent panels. This is important for determining the next element in the streamline trace when a panel boundary is encountered. It is also used to determine groups of elements to start streamlines when the elements are on different panels. The tolerance input is for the maximum gap that may exist in the geometry. The code uses an adjustable gap tolerance based on the input value. This logic allows streamlines to jump large gaps while not missing small elements.

**Geometry Gap Tolerance:**

**Figure 3.1-4 Geometry Gap Tolerance Input**

**3.1.3.2 Symmetry** - The geometry symmetry option is shown in Figure 3.1-5 and is used to specify the type of geometry that will be used for the streamline calculation. The default value is "Half Geometry", which signifies that only the left side of the geometry is input. If a sideslip angle is entered in conjunction with the "Half Geometry" option, all streamlines will stop at the plane of symmetry. The "Full Geometry" option is used if both the left and right sides of the geometry are specified. This option must be used to obtain streamlines on both sides of the vehicle.

**Geometry Symmetry:**

Half Geometry	<input checked="" type="checkbox"/>
Full Geometry	<input type="checkbox"/>

**Figure 3.1-5 Geometry Symmetry Option**

**3.1.3.3 Streamline Direction** - The streamline direction option is shown in Figure 3.1-6 and allows the user to trace streamlines from Back-to-Front or from Front-to-Back. The default direction is from Back-to-Front. This is the preferred option since all streamlines start at different locations and converge to a single point, which allows for better streamline coverage.

Streamline Direction: 

Back-to-Front	<input checked="" type="checkbox"/>
Front-to-Back	<input type="checkbox"/>

**Figure 3.1-6 Streamline Direction Option**

**3.1.3.4 Streamline Definition** - The streamline is defined by points calculated at the boundaries of elements. The streamline definition as calculated by the program contains at most three points per element. The first point is defined where the streamline enters the element, the second point is calculated on the interior of the element, and the third point is where the streamline exits the element. For a complete discussion on the streamline calculation process see Reference 5, QUADSTREAM User's/Reference Manual. Note: In actuality the exit point of one element is the entrance point for the next element and the program only stores the one point.

The streamline definition options are shown in Figure 3.1-7 and allow the user to control the output generated from QUADSTREAM. The "Two Points per Element" option is the default option and produces the minimum number of streamline points while maintaining the accuracy of the streamline trace. The "Three Points per Element" option is similar to the "Two Points per Element" option but includes any interior points. This is the most accurate option since the output is identical to the results of the streamline calculation. The "Evenly Distributed" option gives the user the capability to space the streamline points equally along the streamline. This option is useful for large elements to distribute the points better. If the "Evenly Distributed" option is chosen, the user must enter the number of points to distribute along each streamline.

Streamline Definition: 

Two Points per Element
Three Points per Element
Evenly Distributed Along Streamline

☒

Number of Points per Streamline: 

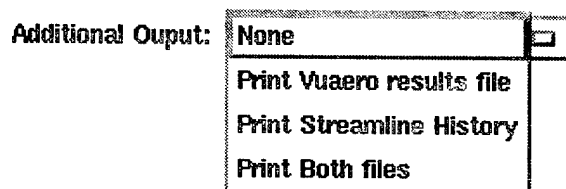
100
-----

**Figure 3.1-7 Streamline Definition Options**

**3.1.3.5 Additional Output** - QUADSTREAM by default writes a file that contains the streamline data that is used by S/HABP. This file is also used to view the streamlines in the geometry modeler. The default streamline file has a ".qstr" extension and is written every time QUADSTREAM is run. The user has the capability to write out two additional files: VUAERO

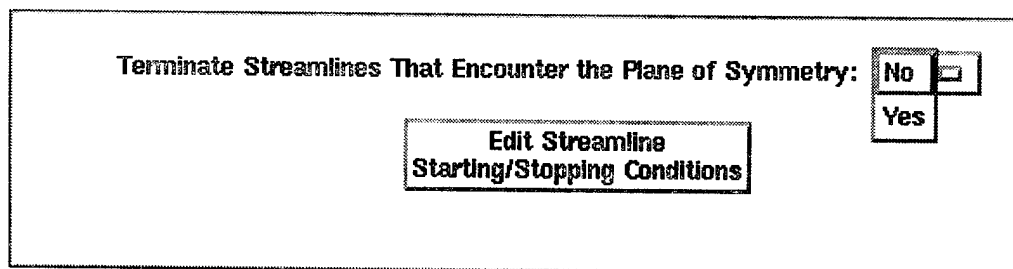
Results file and Streamline History file. The VUAERO Results file contains the x,y,z coordinates for the streamline trace and is formatted to be used with the VUAERO program. Refer to Reference 3, VUAERO Reference Manual, for a description of the VUAERO code. A new VUAERO Results file is written for each alpha-beta combination since VUAERO is limited to a single flight condition per file. The VUAERO file has a ".qv.##" extension, where the ## represents a number from 1-99 and is incremented for each alpha-beta combination. The Streamline History file contains a element by element trace of each streamline. The file also includes error/warning messages for each streamline. This file is useful when trying to debug problems with the streamline trace. The file has a ".qhis" extension.

The additional output option is shown in Figure 3.1-8 and allows the user to write additional files. Either the VUAERO Results file or the Streamline History file can be written by selecting the appropriate menu item. Both files can be printed by selecting the "Print Both files" menu item.



**Figure 3.1-8 Streamline Output Options**

**3.1.3.6 Streamline Starting/Stopping Conditions** - Figure 3.1-9 shows the streamline starting/stopping conditions dialog area on the QUADSTREAM main window. This area gives the user access to the streamline starting/stopping conditions window. Press the "Edit Streamline Starting/Stopping Conditions" button to bring up the Streamline Starting/Stopping Conditions window. The Streamline Starting/Stopping Conditions window will be discussed in Section 3.1.4. The remaining option in this dialog area allows the user to terminate streamlines at the plane of symmetry. The default selection is "No" which allows streamlines that encounter the plane of symmetry to continue to the stagnation point. Select "Yes" to terminate streamlines at the point where the streamline reaches the plane of symmetry. This option is useful for high angles of attack where the flow along the body may not originate at the nose.



**Figure 3.1-9 Streamline Starting/Stopping Options**

### 3.1.4 Streamline Starting/Stopping Conditions

The streamline starting/stopping conditions window is shown in Figure 3.1-10 and is used to enter the streamline starting and stopping conditions for each QUADSTREAM part. Only parts that have components associated with them are available to start and stop streamlines. The part name is shown at the top of the window in the option menu labeled Current QUADSTREAM part. This option menu is used to switch the current part. Before switching parts, the user must click the "APPLY" button to record any inputs entered for the current part.

**NOTE:** All inputs displayed and entered are for the current part displayed at the top of the window.

**Current Quadstream Part:** Wing ☐

---

**Streamline Termination Options**

**Part Tracing Option:** Streamlines Terminate on Current Part ☐

**Stop Streamlines at:** Fwd. Most Point on Part ☐

X-Station

---

**Streamline Starting Position Options**

**Distributed Starting Options:** Evenly Distributed at T.E. ☐

**Number of Starting Points:** 5

X-Station:

---

**Panel ID, Element ID, Number of Streamlines**

1, 120, 5
1, 121, 5
1, 130, 2

**X,Y,Z Starting Locations**

-150.0, 0.0, -5.0
-150.0, 5.0, 0.0
-150.0, 0.0, 5.0

OK Apply Cancel Help

Figure 3.1-10 Streamline Starting/Stopping Conditions Window



**3.1.4.1 Distributed Starting Options** - The distributed starting options are shown in Figure 3.1-11 and are the fastest and easiest way to start streamlines on a part. There are two basic means to distribute streamlines on a part. The first way is to equally space a specified number of streamlines at a given cross section. The second way is to start a specified number of streamlines on each element at a given cross section.

**None:** This option is used if the user does not want to use the distributed streamline options.

**Evenly Distributed at T.E.:** This option allows the user to start a group of streamlines equally spaced at the trailing edge of a part. The user can specify the number of streamlines to start at the trailing edge by entering the number in the "Number of Starting Points" field. The "X-Station" field is grayed out because no value is required.

**At Each T.E. Element:** This option allows the user to start a group of streamlines on each trailing edge element on a part. The user can specify the number of streamlines to start on each element by entering the number in the "Number of Starting Points" field. The "X-Station" field is grayed out because no value is required.

**Evenly Distributed at X-Station:** This option allows the user to start a group of streamlines equally spaced at a user specified x-station. The user can specify the number of streamlines to start at the x-station by entering the number in the "Number of Starting Points" field. The user can specify the x-station by entering the desired x-coordinate in the "X-Station" field.

**At Each Element at X-Station:** This option allows the user to start a group of streamlines on each element at a user specified x-station. The user can specify the number of streamlines to start on each element by entering the number in the "Number of Starting Points" field. The user can specify the x-station by entering the desired x-coordinate in the "X-Station" field.

**Note:** If the "Evenly Distributed" options fail, the user can still use the "At Each Element" options to start groups of streamlines on a part.

**Distributed Starting Options:**

None
Evenly Distributed at T.E.
At Each T.E. Element
Evenly Distributed at X-Station
<b>At Each Element at X-Station</b> <input checked="" type="checkbox"/>

**Number of Starting Points:**

**X-Station:**

**Figure 3.1-11 Distributed Starting Options**

**3.1.4.2 Element Starting Option** - The element starting dialog shown in Figure 3.1-12, allows the user to enter up to 30 elements to start streamlines. Each element is entered on a separate line and is identified using the panel identification number and the element identification number from the geometry file. The panel identification number is entered first, then the element identification number followed by the number of streamlines to be started on each element. All entries on a line are separated by a space or comma. Additional elements can be entered anywhere in the list. Elements can be deleted by deleting the values on a single line (Note: The blank line will remain until the OK button is clicked.). Modifications can be made to an entry by moving the pointer to the line displaying the entry desired and editing the values.

**NOTE:** The panel and element identification numbers can be found from the geometry modeler. First, the user turns on the "Display Centers" option under the Display pull down menu. Then the user clicks with the middle mouse button on the desired element centroid which is shown in red. Use shift/middle mouse to pick multiple elements. The panel and element identification numbers are displayed at the bottom of the window.

**Panel ID, Element ID,  
Number of Streamlines**

1, 120, 5
1, 121, 5
1, 130, 2

**Figure 3.1-12 Element Starting Option**

**3.1.4.3 X,Y,Z Starting Option** - The x,y,z starting dialog shown in Figure 3.1-13, allows the user to enter up to 30 points to start streamlines. Each point is entered on a separate line using the desired coordinates from the geometry. The x-coordinate is entered first, then the y-coordinate followed by the z-coordinate. The coordinates are separated by a spaces or commas. Additional points can be entered anywhere in the list. Points can be deleted by deleting the values on a single line (Note: The blank line will remain until the OK button is clicked.). Modifications can be made to an entry by moving the pointer to the line displaying the entry desired and editing the values.

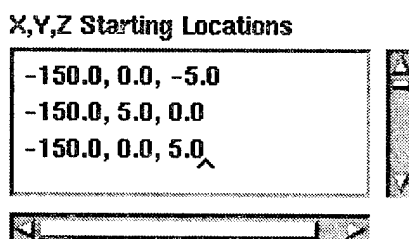


Figure 3.1-13 X,Y,Z Starting Option

**3.1.4.4 Termination Options** - In addition to starting streamlines the user must enter the stopping conditions for each QUADSTREAM part. The stopping conditions are used to stop sets of streamlines on a specified part.

**Part Tracing Option:** The part tracing option shown in Figure 3.1-14 is used to control whether or not streamlines trace onto other QUADSTREAM parts. Selecting “Streamlines Terminate on Current Part” terminates all streamlines on the current part at the specified stopping condition. Selecting “Streamlines Allowed to Trace on Other Parts” allows streamlines that encounter another part before reaching the specified stopping condition to continue onto the adjacent part.

**NOTE:** Streamlines follow the stopping conditions for the part the streamline is currently on. For example, If a streamline was started on the wing and traced onto the body, the streamline would terminate at the stopping condition for the body not the wing.

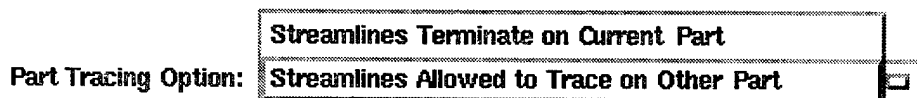


Figure 3.1-14 Part Tracing Option

**Streamline Termination Option:** The streamline termination option gives the user the capability to stop sets of streamlines at different points on the vehicle. The termination options are dependent on the current QUADSTREAM part.

Figure 3.1-15 shows the streamline termination option menu for Body and Inlet parts. The stagnation element option is only valid for the Body part and terminates streamlines at an element perpendicular to the velocity vector. The stagnation element is calculated by checking the normals of each element relative to the velocity vector. The element that is closest to being perpendicular and within a preset tolerance is used as the stagnation element. If no elements are found within the preset tolerance, streamlines are terminated at the leading edge of the part. The "L.E. of Part if Encountered" option is valid for both Body and Inlet parts and terminates streamlines at the forward most point on the part. The "X-Station" option is valid for both Body and Inlet parts and terminates streamlines at a user specified x-station. The user specifies the x-station in the X-Station input field.

Stop Streamlines at:	Stagnation Element
	L.E. of Part if Encountered
	X-Station <input type="checkbox"/>
	X-Station 0.00

**Figure 3.1-15 Streamline Termination Option**

Figure 3.1-16 shows the streamline termination option menu for Wing, Fin and Tail parts. The "L.E. of Part if Encountered" option is used to stop streamlines at the first point where the streamline reaches the leading edge. This option is useful for wings with leading edge sweep where the flow does not originate at the root. The leading edge point where the streamlines terminate are based on the change in streamline direction. If the relative change in streamline direction exceeds a preset parameter, the streamline terminates. This parameter can be changed by editing the vecc.par file and recompiling the code. The "Fwd. Most Point on Part" option is useful for wings with leading edge sweep where the flow originates at the root. This option terminates streamlines at the forward most point on the part. The "X-Station" option is the same as for Body and Inlet parts.

Stop Streamlines at:	L.E. of Part if Encountered
	Fwd. Most Point on Part <input type="checkbox"/>
	X-Station

**Figure 3.1-16 Streamline Termination Option For Wing, Fin And Tail Parts**

## 3.2 Applicability for Complex Configurations

This section discusses general guidelines for running QUADSTREAM and discusses some inherent problems such as streamline coverage and premature streamline termination.

### 3.2.1 General Guidelines

QUADSTREAM is capable of tracing streamlines on complex configurations with multiple components; however, there are some simple guidelines to follow when defining the geometry and setting up the streamline inputs that will ensure satisfactory streamline traces. See also the Error/Warning Messages Section for indications of any of the following problems.

When building a geometry to calculate streamlines, the user should keep in mind several considerations that affect the streamline calculation including gaps; size and shapes of elements; accuracy of quadrilateral representation and component definition. The user should refer to Reference 5, QUADSTREAM User's/Reference manual, for more information about the methods used to calculate streamlines.

**Gaps:** Gaps occur at the juncture between panels or sections primarily for two reasons. The first reason is that the panels were not built very accurately and second because the two adjacent panels have a different number of elements per cross sections or a different number of cross sections. QUADSTREAM was designed to handle most gaps; however, to minimize problems related to gaps the user should minimize any gaps in the geometry.

**Elements:** Much consideration needs to be given at the element level due to the fact that QUADSTREAM is an element based streamline code. The size in conjunction with the shape of an element can cause several problems during the streamline calculation. QUADSTREAM uses vector math and trigonometric relations to calculate the streamline trace which are sensitive to the numerical accuracy of the geometry definition.

Very small elements which the relative distance between any two nodes is on the order of magnitude of the accuracy of the geometry description can cause problems in the streamline calculation. Also elements that are much larger in one direction than the other direction can also cause problems. The first problem can be solved by scaling the entire geometry or increasing the size of the small elements. The second problem can be solved by breaking the element up into smaller more proportionate size elements.

Element twist can cause streamlines to terminate prematurely because of a conflict in streamline direction and should therefore be kept to a minimum. To solve this problem, the twisted element can simply be divided up into smaller elements.

Degenerate quadrilateral elements present another problem. Degenerate elements can exist as triangular and bar elements. Bar elements are elements with zero area and are discarded during the geometry preprocessing stage. Triangular elements can cause problems because more than two elements can share a single node, which complicates the calculation of the streamline direction. QUADSTREAM attempts to determine the correct path; however, problems can arise due to conflicts in streamline direction.

**Quadrilateral Representation:** QUADSTREAM uses a quadrilateral based geometry to calculate the streamline trace. However, the quadrilateral elements only approximate the actual surface as shown in Figure 3.2-1. It should be kept in mind that the more elements used to build the geometry the closer the approximation will be to the actual geometry. The biggest problem associated with a geometry that is defined with only a few elements is conflicting streamline directions at element boundaries. A conflict in streamline direction can occur because QUADSTREAM uses only the normals of each element and not the actual surface normal. A conflict occurs when the streamline direction of one element conflicts with that of the adjacent element. If a conflict does occur, the streamline traces along the boundary between the conflicting elements. This is normally not a problem, but can lead to some strange looking streamlines on geometries with large elements.

Calculating the stagnation element can be another problem due to the lack of enough elements. The stagnation element is calculated based on the slope of the element relative to the velocity vector. There is a preset tolerance of 10 degrees that is used to determine the stagnation element. If no elements are found within this tolerance, QUADSTREAM uses the leading edge to stop streamlines.

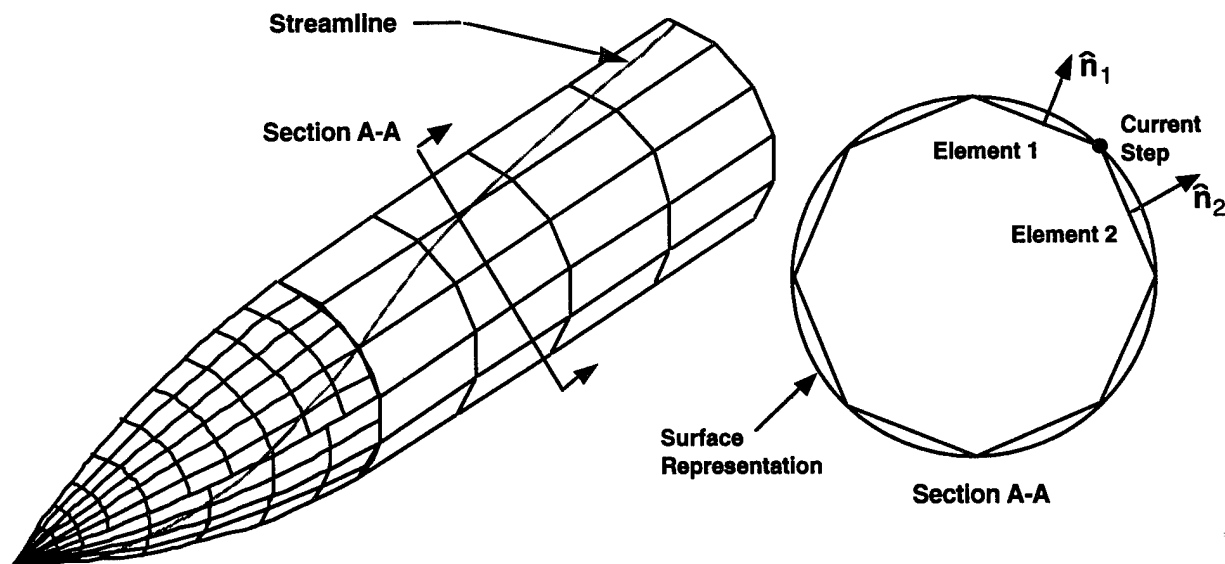
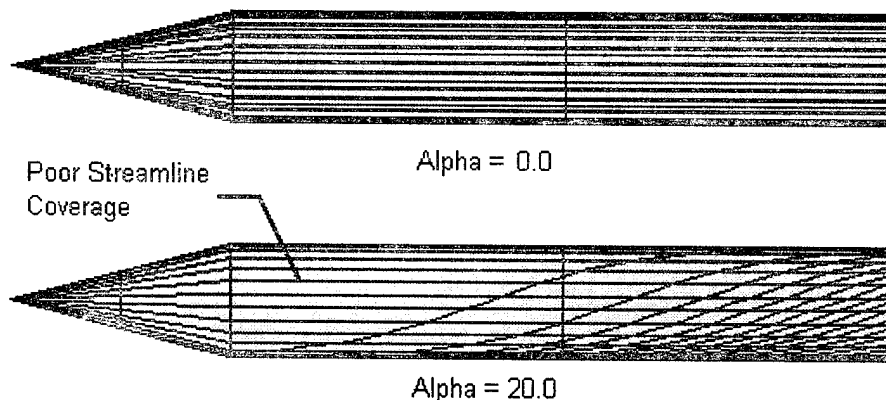


Figure 3.2-1 Surface Versus Quadrilateral Representation

**Component Definition:** Component definition is important since components are used to start and stop sets of streamlines. Component definition can be divided into two categories: Component grouping, and Panel definition. In the QUADSTREAM program the user defines parts based on the components defined in the geometry modeler. Streamlines are then started and stopped on these parts. The user can group components selectively to achieve satisfactory coverage. The user can also create panels to achieve a certain end result. For example, if the user wanted streamlines to trace along a highly swept leading edge, the leading edge should be clearly defined. In contrast, if a user wanted streamlines to terminate at the leading edge, the leading edge could be defined by a line which would cause all streamlines to stop at the point where they reached the leading edge. There are program options to achieve the same results; however, manipulating the geometry can assure the desired results.

### 3.2.2 Streamline Coverage

Streamlines are usually calculated to be used with S/HABP analyses. It is important to have good streamline coverage for S/HABP analyses since the properties calculated along the streamline must be interpolated to each element for output purposes and force and moment summation. The number of streamlines and the distribution of streamlines required to produce satisfactory results vary with each case. It is recommended that the user calculate streamlines on the geometry, view the streamlines, and then add more streamlines if necessary before running S/HABP. Streamlines can be added anywhere on the geometry using the Element ID or XYZ options to achieve satisfactory streamline coverage. It should be noted that as the angle-of-attack increases, streamlines tend to converge near the bottom centerline, leaving the top of the vehicle sparsely covered as illustrated in Figure 3.2-2. This problem can usually be solved by adding more streamlines near the top centerline closer to the front of the vehicle which gives a much better distribution of streamlines over the surface of the vehicle.



**Figure 3.2-2 Streamline Coverage At Angle-Of-Attack.**

### 3.2.3 Premature Stopping of Streamlines

Premature stopping of streamlines refers to streamlines that terminate before reaching the designated stopping condition. This is important since streamlines are usually calculated for use with S/HABP analyses such as flowfield and level 2 viscous analysis. S/HABP level 2 viscous analysis calculates aeroheating and boundary layer properties along the streamline. It assumes that the first point of the streamline corresponds to the stagnation point. If a streamline stopped prematurely, the running lengths along the streamline would be erroneous and the resultant boundary layer properties and aeroheating data would be wrong. The S/HABP flowfield analysis calculates shocks and resultant flowfield properties along the streamline. It assumes that the first point of the streamline is at the leading edge of the vehicle. If a streamline stopped prematurely, a shock would be generated at the stopping point and therefore the resulting flowfield properties would be wrong. The user should always view the resultant streamlines before running S/HABP.

### 3.3 Error/Warning Messages

This section lists and describes all the error and warning messages generated by QUADSTREAM during program execution. In general, all errors will terminate program execution, and all warnings will terminate the current process and continue execution.

**NOTE:** Panel IDs/Element IDs referenced to by all error/warning messages refer to the panel ID/element ID from the VECC geometry.

#### 3.3.1 User Interface Error Message

All the messages in this section refer to errors/warnings that occur while entering values in the GUI. The warning messages are to inform the user that a limit has been exceeded or the entry is invalid. None of these warnings will terminate program execution; however, the invalid inputs must be changed before execution can continue. The error dialog shown in Figure 3.3-1 is displayed if any of the streamline inputs are invalid. Information on the specific error is contained in the message area at the bottom of the Streamline Input window.

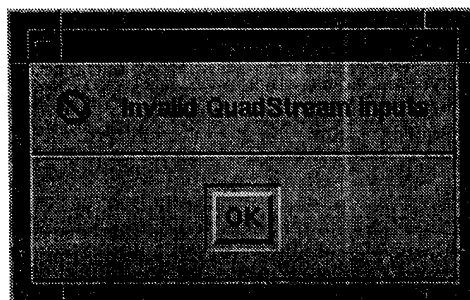


Figure 3.3-1 QUADSTREAM Error Dialog.



**\*\*WARNING\*\*** Starting point number 10 does not correspond to point on selected geometry.

This warning message is given if the program is unable to determine a point on the geometry from the coordinates entered. If this occurs, the program skips the point and continues execution.

**\*\*WARNING\*\*** Starting point number 10 is not on current QUADSTREAM part.

This warning message is given if the point found on the geometry does not lie on the current QUADSTREAM part. If this occurs, the program skips the point and continues execution.

**\*\*WARNING\*\*** Panel# 2, Element# 100 does not correspond to selected geometry.

This message is given if the Panel ID and Element ID entered are greater than the maximum IDs for the part. If this occurs, the program skips the element and continues execution.

**\*\*WARNING\*\*** Panel# 2, Element# 100 is not on current QUADSTREAM part.

This message is given if the Panel ID and Element ID entered is not on the current part. If this occurs, the program skips the element and continues execution.

**\*\*WARNING\*\*** Unable to locate stagnation element within MAXIMUM angular error. Streamlines will terminate at L.E. of Body.

This warning message is given if a stagnation element cannot be determined within an acceptable tolerance. If this occurs all streamlines will terminate at the leading edge of the body.

**\*\*WARNING\*\*** Unable to determine starting positions (encountered another part) Problem occurred around Element# 1 of Panel# 2.

This message is given when the default routine tries to locate the group of elements to distribute the streamlines and one of the elements does not belong to the current part.

**\*\*WARNING\*\*** Unable to determine starting positions (gap size larger than tolerance) Problem occurred around Element# 1 of Panel# 2.

This error message is given when the default routine tries to determine the congruency between a group of elements and cannot locate an element due to the gap between the elements being too large.

**\*\*WARNING\*\*** Unable to determine trailing edge elements

This error is given when trying to distribute streamlines on wing type components and the program cannot determine the trailing edge elements.

**\*\*ERROR\*\*** No streamlines have been entered !

This error is given if no streamlines have been entered and the user tries to start execution.

**\*\*WARNING\*\*** Total number of streamlines exceeds MAXIMUM (100) !

This error is given if the total number of streamlines exceeds the maximum set in the vecc.par file. If this message occurs, the user must eliminate streamlines to reduce the total number of streamlines below the maximum.

**\*\*ERROR\*\*** Distributed Starting Option Failed !

This error is given if one of the distributed starting options failed. The user must modify the streamline inputs before continuing execution. Information on the specific error is contained in the message area at the bottom of the Streamline Input window.

**\*\*WARNING\*\*** Some or All starting points have been eliminated from set  
!

This warning is given if any points started using the XYZ starting option were eliminated because of errors. Information on the specific reason for the points being eliminated is contained in the message area at the bottom of the Streamline Input window.

**\*\*WARNING\*\*** Some or All Elements have been eliminated from set !

This warning is given if any points started using the Element ID option were eliminated because of errors. Information on the specific reason for the streamlines being eliminated is contained in the message area at the bottom of the Streamline Input window.

**\*\*ERROR\*\*** Must input stopping value when tracing FRONT to BACK !

This error is given if the user attempts to terminate the streamlines using the leading edge option, stagnation point option or forward most point option when tracing front to back. The user must enter an x-station to terminate the streamlines if tracing from front to back.

**\*\*ERROR\*\*** Cannot distribute streamlines at T.E. when tracing FRONT to  
BACK !

This error is given if the user attempts to distribute streamlines on wing type parts when tracing front to back.

### 3.3.2 I/O File Error Messages

This section describes error messages that are given during opening or reading files.

**\*\*ERROR\*\*** Error opening default streamline file. (.qstr)

This error is given if the program is unable to open the streamline data output file. If this occurs, QUADSTREAM terminates and control is returned to the GUI.

**\*\*ERROR\*\*** Error opening VUAERO streamline file. (.qv.##)

This error is given if the program is unable to open the data file to write the VUAERO data file. If this occurs, QUADSTREAM terminates and control is returned to the GUI.

**\*\*ERROR\*\*** Error opening QUADSTREAM history file. (.qhis)

This error message is given if the program is unable to open QUADSTREAM history file. If this occurs, QUADSTREAM terminates and control is returned to the GUI.

### 3.3.3 Streamline Calculation Error Messages

The messages discussed in this section deal with errors during the streamline calculation and can occur throughout the program.

**\*\*WARNING\*\*** Maximum number of points per streamline reached.  
Current Maximum = 500

This warning message is given if the number of streamline points for the current streamline reaches the limit. If this occurs the streamline trace will terminate at the current streamline position. The current limit is shown above and is printed out when the error occurs.

**\*\*WARNING\*\*** Streamline stopped because it switched axial direction.

This message is given if a streamline tries to trace in the opposite axial direction. This should only occur near the stagnation point and is considered normal termination.

**\*\*WARNING\*\*** Streamline encountered another part and stopped.

This message is given if a streamline tries to trace onto part and the Part-to-Part tracing is turned off. If this occurs the streamline trace will terminate.

**\*\*WARNING\*\*** Element# 100 on Panel# 2 is too large relative to gap size.

This warning message is given if an element is too large relative to the gap tolerance. This occurs when the program is attempting to identify an element neighbor. The program steps across each element using an increment based on the size of the element; however, if the increment is larger than the gap size, the closest point may not be within the gap tolerance. This problem can be remedied by increasing the gap tolerance or by subdividing the element.

**\*\*WARNING\*\*** Streamline stopped due to gap larger than tolerance.

This message is given if the program is unable to determine the next element at a panel boundary due to the closest element being further than the gap tolerance. This problem can be eliminated by increasing the gap tolerance

**\*\*ERROR\*\*** Unable to determine group of elements at X = -10.00.

This message is displayed if the program is unable to determine a set of elements at the specified x-station. The program looks for a group of elements at an x-station equal to the printed value plus/minus the gap tolerance depending on which way streamlines are being traced. If the gap tolerance is too large, the calculated x-station can skip a set of elements or even be off the geometry.

**\*\*WARNING\*\*** Streamline encountered a node on Element# 100 of Panel# 2 and stopped because no adjacent element was found to determine the streamline direction.

This warning message is given if the current streamline position is at a node and the program could not locate an adjacent element which is needed to determine the streamline direction. If this occurs, the streamline trace will terminate.

**\*\*WARNING\*\*** Unable to determine if current position is at the plane of symmetry. Problem occurred on Element# 100 of Panel# 2.

This warning message is given if the current streamline position is at the plane of symmetry but the code could not determine if the streamline was at the top centerline or the bottom centerline. This information is required to determine if the streamline trace should be terminated. If this occurs, the streamline trace will terminate.

**\*\*WARNING\*\*** Divide by ZERO, Probably due to incorrect Panel orientation. Error occurred on Element# 100 of Panel# 2.

This warning message is given if the value of the denominator in a calculation is zero. The most probable cause for this error is that the Panel orientation is incorrect.

**\*\*WARNING\*\*** Unable to determine direction of streamline

Element# 1 of Panel# 2 is much larger in one direction than the other and may need to be subdivided

This message is given when the streamline calculation fails and the resultant streamline position does not lie on the current element. The element identified was determined to be at least an order of magnitude larger in one direction than the other. A possible solution is to subdivide the element.

**\*\*WARNING\*\*** Unable to determine direction of streamline  
Element# 1 of Panel# 2 is very small and could have caused numerical accuracy problems.

This message is given when the streamline calculation fails and the resultant streamline position does not lie on the current element. The problem is usually due to numerical accuracy. The average length of the element identified was determined to be only an order of magnitude larger than the GEOMTOL parameter. A possible solution is to scale the entire geometry.

**\*\*WARNING\*\*** Unable to determine direction of streamline  
Problem has occurred around Element# 1 of Panel# 2  
Refer to User Manual for additional information.

This message is given when the streamline calculation fails and the resultant streamline position does not lie on the current element. The problem is usually due to numerical accuracy. QUADSTREAM was unable to determine a possible reason for the failure using an order of magnitude analysis. The element identified is probably either very small or much larger in one direction than the other.

**\*\*WARNING\*\*** Streamline stopped due to Element# 100 of Panel# 2 is twisted. A conflict in streamline direction occurred at the intermediate side.

This warning message is given when a streamline encounters the intermediate side of an element and stops due to a conflict in streamline direction. The problem usually arise due to the element being twisted. Unlike conflicts in direction at element boundaries, conflicts at the intermediate side cannot be resolved without introducing a "jump" or "kink" in the streamline. To eliminate this problem the user can either untwist the element or divide the element which effectively reduces the amount of twist.

**\*\*WARNING\*\*** Streamline traced back onto last element.  
Panel ID = 2, Element ID = 100

This warning message is given if a streamline tries tracing back onto the last element. If this occurs the streamline trace will be terminated. This problem usually occurs near the leading edge of a component and is due to the high curvature and relative incidence angle of the element to the velocity vector.

### 3.4 Output Files

QUADSTREAM by default writes a file that contains the streamline data that is used by S/HABP. This file is also used to view the streamlines in the geometry modeler. The default streamline file has a ".qstr" extension and is written every time QUADSTREAM is run. The user has the capability to write out two additional files: VUAERO Results file and Streamline History file. See Section 3.1.3.5 for instructions on how to obtain the additional output. The VUAERO Results file contains the x,y,z coordinates for the streamline trace and is formatted to be used with the VUAERO program. A new VUAERO Results file is written for each alpha-beta combination since VUAERO is limited to a single flight condition per file. The VUAERO file has a ".qv.##" extension, where the ## represents a number from 1-99 and is incremented for each alpha-beta combination. The Streamline History file contains a element by element trace of each streamline. The file also includes error/warning messages for each streamline. This file is useful when trying to debug problems with the streamline trace. The file has a ".qhis" extension

This section contains samples of each file generated for the AFWAL Elliptic Body. Streamlines generated on the AFWAL Elliptic Body at 0 degrees angle-of-attack are shown in Figure 3.4-1.

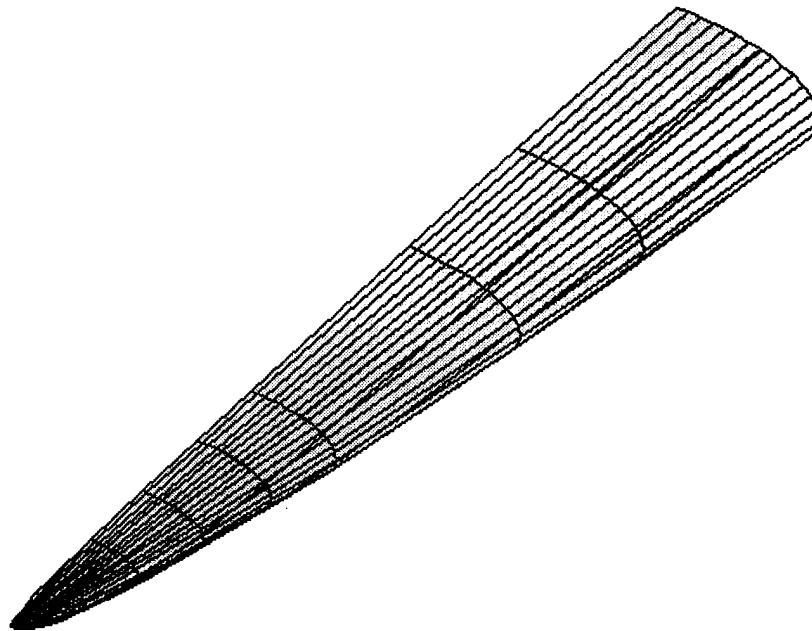


Figure 3.4-1 QUADSTREAM Generated Streamlines On AFWAL Elliptic Body (Alpha = 0.0).

### 3.4.1 QUADSTREAM Default Streamline File (.qstr)

The default streamline file written by QUADSTREAM contains the x,y,z coordinates for the streamline trace, the geometry unit normal at each streamline point, the wetted distance along the streamline and the component ID at each point as illustrated in Figure 3.4-2. The default streamline file contains the streamline data for all alpha-beta combinations. The filename uses the base filename for the configuration with a ".qstr" extension. This file is written every time QUADSTREAM is run. The ".qstr" file is used for level 2 viscous and flowfield analysis in S/HABP. In addition the file is used to view streamlines in the geometry modeler. A sample output file containing streamline data for five streamlines on the AFWAL Elliptic Body at two angles-of-attack is shown in Figure 3.4-3

```

write(iunit,*) num_alpha_beta
do iab = 1, num_alpha_beta
  write(iunit,*) alpha(iab), beta(iab)
enddo

do iab = 1, num_alpha_beta
  write(iunit,*) alpha(iab), beta(iab)
  do istrm = 1, num_strm(iab)
    write(iunit,*) num_pts(iab,istrm)
    do ipt = 1, num_pts(iab,istrm)
      write(iunit,*) x(ipt), y(ipt), z(ipt), nx(ipt), ny(ipt), nz(ipt), wet_dist, compID
    enddo
  enddo
enddo

```

Where:

<b>num_alpha_beta</b>	number of alpha/beta combinations
<b>alpha</b>	angles-of-attack
<b>beta</b>	sideslip angles
<b>num_strm</b>	number of streamlines for each flight condition
<b>num_pts</b>	number of points/streamline
<b>x</b>	x-coordinates for each streamline
<b>y</b>	y-coordinates for each streamline
<b>z</b>	z-coordinates for each streamline
<b>nx</b>	x component of the unit normal for each streamline
<b>ny</b>	y component of the unit normal for each streamline
<b>nz</b>	z component of the unit normal for each streamline
<b>wet_dist</b>	wetted distance along the streamline
<b>compID</b>	S/HABP component identification number of each point

**Figure 3.4-2 QUADSTREAM Default Output File Format.**

```

2
    0.0000    0.0000
    10.0000    0.0000
    0.0000    0.0000    5
13
0.0000    0.0000    0.0000    0.5049    0.0140    -0.8631    0.0000    1
-1.0000    0.0000    -0.5850    0.2444    0.0155    -0.9696    1.1585    1
-2.0000    0.0000    -0.8370    0.2443    0.0155    -0.9696    2.1898    1
-3.5000    0.0000    -1.2150    0.1549    0.0159    -0.9878    3.7367    1
-7.0000    0.0000    -1.7640    0.0963    0.0161    -0.9952    7.2795    1
-11.0000    0.0000    -2.1510    0.0815    0.0161    -0.9965    11.2982    1
-22.0000    0.0000    -3.0510    0.0616    0.0161    -0.9980    22.3349    1
-36.0000    0.0000    -3.9150    0.0515    0.0161    -0.9985    36.3616    1
-51.0000    0.0000    -4.6890    0.0539    0.0161    -0.9984    51.3815    1
-65.5000    0.0000    -5.4720    0.0362    0.0162    -0.9992    65.9026    1
-109.0000    0.0000    -7.0470    0.0291    0.0161    -0.9994    109.4312    1
-139.0000    0.0000    -7.9200    0.0245    0.0161    -0.9996    139.4438    1
-183.0000    0.0000    -9.0000    0.0245    0.0161    -0.9996    183.4571    1
.
.
.
10.0000    0.0000    5
13
0.0000    0.0000    0.0000    0.5049    0.0140    -0.8631    0.0000    1
-1.0000    0.0000    -0.5850    0.2444    0.0155    -0.9696    1.1585    1
-2.0000    0.0000    -0.8370    0.2443    0.0155    -0.9696    2.1898    1
-3.5000    0.0000    -1.2150    0.1549    0.0159    -0.9878    3.7367    1
-7.0000    0.0000    -1.7640    0.0963    0.0161    -0.9952    7.2795    1
-11.0000    0.0000    -2.1510    0.0815    0.0161    -0.9965    11.2982    1
-22.0000    0.0000    -3.0510    0.0616    0.0161    -0.9980    22.3349    1
-36.0000    0.0000    -3.9150    0.0515    0.0161    -0.9985    36.3616    1
-51.0000    0.0000    -4.6890    0.0539    0.0161    -0.9984    51.3815    1
-65.5000    0.0000    -5.4720    0.0362    0.0162    -0.9992    65.9026    1
-109.0000    0.0000    -7.0470    0.0291    0.0161    -0.9994    109.4312    1
-139.0000    0.0000    -7.9200    0.0245    0.0161    -0.9996    139.4438    1
-183.0000    0.0000    -9.0000    0.0245    0.0161    -0.9996    183.4571    1
.
.
.

```

**Figure 3.4-3 QUADSTREAM Default Output File For AFWAL Elliptic Body.**



### 3.4.2 VUAERO Streamline File (.qvu.##)

The VUAERO Streamline file contains the x,y,z coordinates of the streamline trace calculated by QUADSTREAM. This file is compatible with VUAERO and can be read and displayed using the streamline results option. Figures 3.4-4 and 3.4-5 show the file format and a sample file for the AFWAL Elliptic Body.

```
do istream = 1,num_streamlines
  write(idbout,*) num_pts,((x(pnt),y(pnt),z(pnt)),pnt=1,num_pts)
enddo istream
```

Where:

<b>num_pts</b>	number of points on the streamline.
<b>num_streamlines</b>	number of streamlines.
<b>x</b>	array of x-coordinates along the streamline
<b>y</b>	array of y-coordinates along the streamline
<b>z</b>	array of z-coordinates along the streamline.

**Figure 3.4-4 VUAERO Streamline File Format.**

13	-183.0000	0.0000000E+00	-9.000000	-139.0000
0.0000000E+00	-7.920000	-109.0000	0.0000000E+00	-7.047000
-65.50000	0.0000000E+00	-5.472000	-51.00000	0.0000000E+00
-4.689000	-36.00000	0.0000000E+00	-3.915000	-22.00000
0.0000000E+00	-3.051000	-11.00000	0.0000000E+00	-2.151000
-7.000000	0.0000000E+00	-1.764000	-3.500000	0.0000000E+00
-1.215000	-2.000000	0.0000000E+00	-0.8370000	-1.000000
0.0000000E+00	-0.5850000	0.0000000E+00	0.0000000E+00	0.0000000E+00
21	-183.0000	13.02753	-7.416976	-182.8041
13.02624	-7.411637	-139.0000	12.65679	-6.171674
-129.1097	12.55785	-5.839809	-109.0000	12.29929
-5.134043	-95.44501	12.08242	-4.542103	-71.91850
11.58882	-3.460686	-65.50000	11.40698	-3.149878
-56.68729	11.03462	-2.513398	-51.00000	10.69290
-2.088303	-44.23950	10.30475	-1.605465	-36.00000
9.595551	-1.058721	-32.19185	9.203539	-0.7564908
-22.00000	7.702014	-0.3578081	-15.09980	6.354231
0.0000000E+00	-11.00000	5.497000	0.0000000E+00	-7.000000
4.508000	0.0000000E+00	-3.500000	3.105000	0.0000000E+00
-2.000000	2.139000	0.0000000E+00	-1.000000	1.495000
0.0000000E+00	0.0000000E+00	0.0000000E+00	0.0000000E+00	0.0000000E+00
21	-183.0000	13.02753	7.416976	-182.8041
13.02624	7.411637	-139.0000	12.65661	6.171726
-129.1051	12.55763	5.839707	-109.0000	12.29910
5.134114	-95.44082	12.08214	4.541998	-71.91394
11.58849	3.460586	-65.50000	11.40676	3.149989
-56.68448	11.03424	2.513322	-51.00000	10.69270
2.088467	-44.23629	10.30435	1.605403	-36.00000
9.595425	1.058885	-32.18914	9.203120	0.7564565
-22.00000	7.701975	0.3579540	-15.09666	6.353575
0.0000000E+00	-11.00000	5.497000	0.0000000E+00	-7.000000
4.508000	0.0000000E+00	-3.500000	3.105000	0.0000000E+00
-2.000000	2.139000	0.0000000E+00	-1.000000	1.495000
0.0000000E+00	0.0000000E+00	0.0000000E+00	0.0000000E+00	0.0000000E+00
13	-183.0000	0.0000000E+00	9.000000	-139.0000
0.0000000E+00	7.920000	-109.0000	0.0000000E+00	7.047000
-65.50000	0.0000000E+00	5.472000	-51.00000	0.0000000E+00
4.689000	-36.00000	0.0000000E+00	3.915000	-22.00000
0.0000000E+00	3.051000	-11.00000	0.0000000E+00	2.151000
-7.000000	0.0000000E+00	1.764000	-3.500000	0.0000000E+00
1.215000	-2.000000	0.0000000E+00	0.8370000	-1.000000
0.0000000E+00	0.5850000	0.0000000E+00	0.0000000E+00	0.0000000E+00

**Figure 3.4-5 VUAERO Streamline File For AFWAL Elliptic Body (Alpha = 0.0).**



## 4. S/HABP ANALYSIS

The central module to the VECC analysis system is the upgraded version of the Supersonic/Hypersonic Arbitrary Body Program (S/HABP) dubbed S/HABP Mark V. Input to the S/HABP code itself has changed only slightly. However, the organization of the preparing the input has changed to take advantage of workstation graphics. When preparing input for S/HABP via the Graphical User Interface (GUI), the user will notice that the program input has been split into "case" input and "run" input. Case inputs are those inputs which stay constant through an analysis such as the Mach number and  $\alpha$ - $\beta$  set. Experienced users of S/HABP will recognize the case input as those variables which are set upon each call to the Aerodynamics Program. In addition, summation of predictions from individual runs (analysis of one component) are set up in the case input windows. The "run" input windows are those individual analyses that are to be performed such as an inviscid pressure analysis on the body or a viscous analysis of the fin. These inputs are generated independently of the case input, and as such create a *run pool* from which runs can be selected for each case. This represents a new philosophy for the input to S/HABP which allows a single run (analysis) to be performed in several cases.

The "case" input and "run" input windows are launched from the main GUI window "Edit" menu. Each of these inputs are discussed in detail in the following sections. Although the case inputs are discussed first, it is not necessary that the case inputs be completed first. In fact because of the window environment, either set of inputs can be entered or altered in any order. Of course, some runs must be entered before you can set up the run summations in the case input window.

### 4.1 Case Setup

Choose "S/HABP Case Setup ..." from the main window "Edit" menu to launch the case input window shown in Figure 4.1-1. This window labeled "Case Definition" has three main window panes and a "daughter" window for flight condition input, each of which are discussed in more detail.

**Case Definition**

Case 1  
Case 2  
Case 3

Add Copy Delete

Edit Flight Conditions/Reference Quantities...

Print Component Force Data in Addition to Summation Data: Yes ☐

Echo Inputs to Output File: No ☐

---

**Summations**

Summation 1  
Summation 2  
Summation 3  
Summation 4

Add Copy Delete

**Select Runs for this Summation**

wing,Inviscid, off, -20.0 deg. (pitch)  
wing,Inviscid, off, -10.0 deg. (pitch)  
wing,Inviscid, off, 0.0 deg. (pitch)  
wing,Inviscid, off, 10.0 deg. (pitch)  
body,Inviscid, off, N/A

---

**Select Other Runs to be Executed**

body,Flowfield, off, N/A

OK Apply Cancel Help

Figure 4.1-1 Case Input Window

#### 4.1.1 Case Definition

This section of the case setup window allows the user to change cases already defined or add new cases. Additionally, this window pane contains option menus for selection of optional output and access to the "Flight Conditions/Reference Quantities" window.

**4.1.1.1 Add/Copy/Delete Case** - There are three buttons below the case definition dialogue area as shown in Figure 4.1-1. These buttons control the addition or removal of new cases to the S/HABP input.

Upon entering the "Case Setup" window, the user must first click on the "Add" button to begin definition of the first case. The name of the case can be altered by clicking on the text and

dragging the mouse to highlight the text "Case 1." Case names can be up to 60 characters in length and will appear as titles in the output file.

Additional cases may be defined by clicking on the "Add" or "Copy" buttons. Click on the name of the case whose input you wish to define (this will highlight the case name). The "Copy" button can be very useful for decreasing the amount of input required by the user. For example if a series of cases is to be run with only Mach number changing between each case, the user should complete definition of Case 1 and copy the case. Then the only input that is needed for the new case is to change the Mach number.

**4.1.1.2 Edit Flight Conditions/Reference Quantities** - Click on the text button "Edit Flight Conditions/Reference Quantities..." to launch the input window for flight conditions and reference quantities. The user is reminded that any text field followed by three dots "..." is a button which will launch another window. The input window for reference quantities and flight conditions is discussed in detail in Section 4.1.4.

**4.1.1.3 Print Component Force Data** - This optional menu allows printing of the forces and moments calculated for each component analyzed (i.e., each run) in addition to the force and moment summations.

**4.1.1.4 Echo Inputs To Output File** - This menu gives the user the option of printing the input data (including geometry) to the output file (.out).

## **4.1.2 Summations**

Summations are performed to sum the predictions from several runs. This allows the user to build up the configuration forces and moments in an arbitrary manner. By selecting runs for summation, the user is also specifying which runs will be executed in the selected case (the case highlighted in the top window pane).

To create a summation, the user clicks on the "Add" button below the "Summations" dialogue area. Runs available for summation are shown in the "Select Runs For This Summation" dialogue area. To select a run, click on the run title. Remember to hold down the control button while clicking to select more than one run. By clicking on the first run of a series and then holding down the shift button while clicking on the last run of the series, the user can select an entire group of runs for summation. When more than one summation has been defined, the active summation is highlighted.

### 4.1.3 Select Other Runs To Be Executed

This dialogue area allows the user to execute runs in the selected case which have not been included in a summation. Only the runs which have not been selected as part of a summation are listed in this dialogue area. Runs are selected in the same manner used in defining summations. After a run has been selected, it remains highlighted. This option is useful when a flowfield/shock shape analysis is to be performed and when the user is not interested in summing the force and moment output.

### 4.1.4 Flight Conditions/Reference Quantities

Flight conditions and reference quantities are set for each case via the "Edit Flight Conditions/Reference Quantities ..." text button. The current case for which input is being made is listed at the top of the window. There are four main areas of the "Flight Conditions" window: Reference Quantities, Derivatives, Flight Conditions, and Flight Attitude. Each of these areas are discussed in detail in the following sections. The user is reminded to click "OK" or "Apply" before closing this window in order to save his/her input.

The screenshot shows a software window titled "Flight Conditions" with a dark border. At the top, it says "Case: Case 1". The window is divided into four main sections:

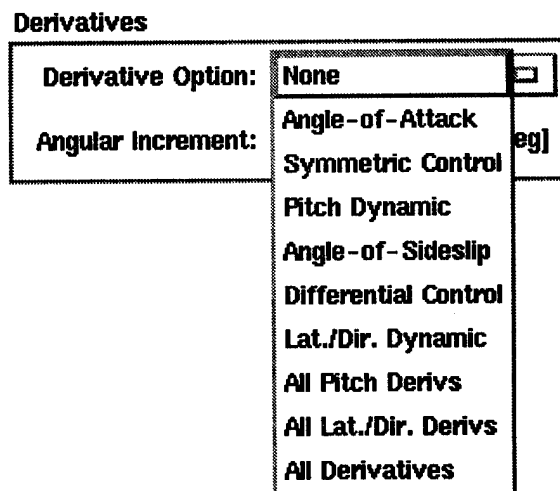
- Reference Quantities:** A box containing input fields for Area (10.000000), Length (100.000000), Span (10.000000), Xcg (-50.000000), Ycg (0.000000), and Zcg (0.000000).
- Derivatives:** A box containing "Derivative Option:" with a dropdown menu set to "None" and a checkbox, and "Angular Increment:" with a text field set to "1.000" and a unit label "[deg]".
- Flight Conditions:** A box containing "Press.Temp. Input:" with a dropdown menu set to "Standard Atmosphere", "Gas Properties:" with a dropdown menu set to "Air", "Mach Number:" with a text field set to "3.000", "Altitude:" with a text field set to "0" and a unit label "[ft]", "Stagnation Pressure:" with a text field and a unit label "[atm]", and "Stagnation Temperature:" with a text field and a unit label "[deg F]".
- Flight Attitude:** A box containing an "Alpha" dropdown menu, a vertical slider for Alpha with a scale from 0.000 to 14.000, and a "Constants" section with input fields for Alpha (0.000), Beta (0.000), Roll (0.000), Pitch Rate (0.000), Yaw Rate (0.000), and Roll Rate (0.000).

At the bottom of the window, there are four buttons: "OK", "Apply", "Cancel", and "Help".

Figure 4.1-2 Flight Conditions Window

**4.1.4.1 Reference Quantities** - The user defines the reference quantities and center of gravity location in this area. The units on the reference quantities must be consistent with the geometry model. Note that center of gravity will usually be a negative number since +X is forward. To enter the input, click in the dialogue area next to the variable name, highlight the default values, use the delete key (or backspace depending on your system) to remove the current values, and then type the desired input values. The variable "Length" is the longitudinal reference length (mean chord), and the variable "Span" is the lateral/directional reference length.

**4.1.4.2 Derivatives** - S/HABP Mark V includes methods for predicting stability and control derivatives. The user selects which type of derivatives are to be predicted for the current case via the option menu labeled "Derivative Option." The available derivatives are shown in Figure 4.1-3 and the corresponding derivatives calculated for each option are listed in Figure 4.1-4.



**Figure 4.1-3 Available Derivative Options**

Option	Derivatives Calculated
None	No derivatives
Angle-of-Attack	$C_{L\alpha}$ , $C_{N\alpha}$ , $C_{m\alpha}$ , $C_{A\alpha}$
Symmetric Control	$C_{N\delta}$ , $C_{m\delta}$ , $C_{L\delta}$ , $C_{Y\delta}$ , $C_{n\delta}$ , $C_{l\delta}$ , symmetric deflection, $\delta_e$
Pitch Dynamic	$C_{Nq}$ , $C_{mq}$ , $C_{Aq}$ , $C_{Nu}$ , $C_{mu}$ , $C_{Au}$ , $C_{N\dot{\alpha}}$ , $C_{m\dot{\alpha}}$
Angle-of-Sideslip	$C_{Y\beta}$ , $C_{n\beta}$ , $C_{l\beta}$
Differential Control	$C_{Y\delta}$ , $C_{n\delta}$ , $C_{l\delta}$ , differential deflection, $\delta_a$ or $\delta_r$
Lat./Dir. Dynamic	$C_{Yp}$ , $C_{np}$ , $C_{lp}$ , $C_{Yr}$ , $C_{nr}$ , $C_{lr}$

**Figure 4.1-4 Derivatives Calculated**

The terms symmetric and differential used here relate to whether the mirrored control surface deflects in the same direction or the opposite direction as the input control surface (remember that only the left side of the vehicle is defined). For example, deflection of a trailing flap is

usually a symmetric control deflection since both the left and right control surfaces deflect in the same direction (i.e. if the left trailing edge is deflected downward then the right flap is mirrored and also deflects downward). In general, rudder and aileron deflections are considered differential deflection. The aileron case is easy to visualize; when one trailing edge deflects downward the other deflects upward. A rudder deflection of twin vertical tails though is also a differential deflection to S/HABP. In this case when the left rudder is deflected trailing edge toward the centerline, the right rudder is deflected trailing edge *away* from the centerline to generate yaw control. Thus the right rudder is not the mirror image of the left rudder and is therefore considered a differential deflection.

The user may also input the angular increment over which angular derivatives are non-dimensionalized. Input is made via the text field next to the label "Angular Increment" and is entered in degrees. The derivative is calculated in S/HABP by running at the input flight conditions, incrementing the flight condition (or control deflection) by the user defined angular increment, recalculating the force and moments, and then using the difference to calculate the stability or control derivative. A default value of 1 degree is automatically selected by the GUI.

Analysis by Moore and Ely (Reference 6) indicates that it is desirable to vary the angle over which the stability and control derivatives are linearized depending on the derivative being calculated. In Reference 6, it was found that using a  $1^\circ$  increment caused unrealistic stability and control design analysis. In their analysis, Moore and Ely used a  $4^\circ$  increment for pitch control derivatives, a  $10^\circ$  increment for aileron derivatives, and a  $20^\circ$  increment for rudder derivatives. Their recommendation is that the derivatives be linearized over the range of expected control surface and flight attitude excursions.

**4.1.4.3 Flight Conditions** - The user defines the case flight conditions in the lower left portion of the "Flight Conditions" window. Flight conditions can be input in one of three ways as shown in Figure 4.1-5. Standard Atmosphere allows the user to specify only the Mach number and altitude with the remainder of the flight conditions being determined via the 1962 Standard Atmosphere table. When this input option is selected, the pressure and temperature input fields are inactive. Using the "Stagnation Properties" option, Mach number, stagnation pressure and stagnation temperature are entered while the altitude field is inactive. Likewise when selecting the "Freestream Properties" option, the freestream pressure and freestream temperature are input in addition to the Mach number and the altitude field is inactive. The other freestream properties are calculated via the isentropic gas equations of NACA 1135. Either air or helium properties may be modeled as selected via the "Gas Properties" option menu.



**Flight Conditions**

Press.Temp. Input:	Standard Atmosphere	
	Stagnation Properties	<input checked="" type="checkbox"/>
	Freestream Properties	
Gas Properties:	Air	<input type="checkbox"/>
Mach Number:	3.000	
Altitude:		[ft]
Stagnation Pressure:	0.0000	[atm]
Stagnation Temperature:	0.0	[deg F]

Figure 4.1-5 Flight Condition Input

**4.1.4.4 Flight Attitude** - The flight attitude input shown in Figure 4.1-6 is essentially the same as the  $\alpha$ - $\beta$  card input to S/HABP. The user can input angle-of-attack (alpha), sideslip angle (beta), roll angle, body axis pitch rate, body axis yaw rate, and body axis roll rate. By default, angle-of-attack is varied while the other inputs remain constant. However, any one of the flight attitude variables can be input as a table by selecting the variable name from the option menu as shown in Figure 4.1-7.

Table input is made one value per line. Input to the table variable (generally angle-of-attack), need not be in ascending order. Additionally, a point within the table can be deleted without re-ordering the table. When the user clicks the "OK" or "Apply" button the table is sorted and blank lines removed.

**Flight Attitude**

<div>Alpha <input checked="" type="checkbox"/></div> <div> ^ 0.000  2.000  4.000  6.000  8.000  10.000  12.000  14.000  v </div>	<b>Constants</b> Alpha: <input type="text"/> Beta: <input type="text" value="0.000"/> Roll: <input type="text" value="0.000"/> Pitch Rate: <input type="text" value="0.000"/> Yaw Rate: <input type="text" value="0.000"/> Roll Rate: <input type="text" value="0.000"/>
--	--

Figure 4.1-6 Flight Attitude Input

**Flight Attitude**

Alpha	Constants
Beta	Alpha: <input type="text"/>
Roll	Beta: <input type="text" value="0.000"/>
Pitch Rate	Roll: <input type="text" value="0.000"/>
Yaw Rate	Pitch Rate: <input type="text" value="0.000"/>
Roll Rate	Yaw Rate: <input type="text" value="0.000"/>
14.000	Roll Rate: <input type="text" value="0.000"/>

**Figure 4.1-7 Any Flight Attitude Variable Can Be Input As A Table**

## 4.2 Run Setup

The VECC GUI uses the idea of a "run pool" to help organize the S/HABP input. This concept allows the user to set up individual analyses for different components independent of the case set up. The run pool can then be used to select runs for summation in any case defined (refer to Section 4.1.2). The user accesses the run setup windows by selecting "S/HABP Run Setup ..." from the main window "Edit" menu. Before beginning run set up, the user must have already grouped panels to form components. After selecting "S/HABP Run Setup ...," the user will enter input in two windows prior to entering data for the specific analysis to be performed. Each of these windows is discussed in detail in Sections 4.2.1 and 4.2.2. The input for specific analyses (such as inviscid pressure analysis) is discussed in Sections 4.3 through 4.6.

### 4.2.1 Run Definition

The first window launched after selecting "S/HABP Run Setup ..." is the "Run Definition" window which is shown in Figure 4.2-1. Any runs which have already been defined will be listed in the center window. The information listed as the run "title" is the component being analyzed, the type of analysis, whether shielding is being performed, and any deflection angle input if applicable. To edit an existing run, click on the run title to select it and click "Edit". This will launch the "Analysis Type/Component" window which is discussed in Section 4.2.2. Runs may be copied and then modified by selecting the run and clicking the "Copy" button, or deleted by clicking the "Delete" button. Pressing "OK" indicates that you are done creating runs and want to return to the main window.

Upon entering the window the first time, the dialogue area will be blank. The user will click "Add" to launch the "Analysis Type/Component" window. Do not click the "OK" button or you will be returned to the main window.

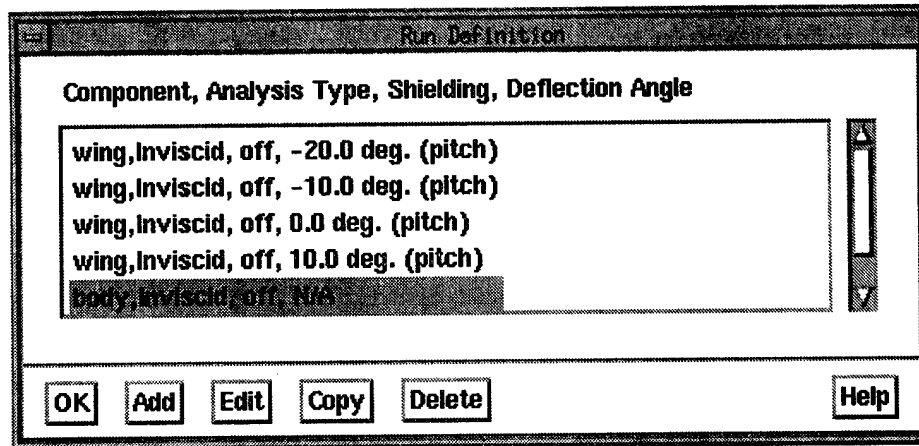


Figure 4.2-1 Run Definition Window

#### 4.2.2 Analysis Type/Component

After choosing to add or edit a run, the "Analysis Type/Component" window is launched. The user selects the type of analysis for this run and the component to be analyzed. If an inviscid pressure analysis or flowfield analysis is selected, all inviscid type components are listed in the "Select Component" dialogue area (refer to Section 2.5 for setting a panels type). If a viscous analysis (either level 1 or level 2) is selected, only viscous components will be listed in the "Select Component" dialogue area. Once the type of analysis and the component are selected press the "OK" button to continue with input.

This window is launched even if the user chose to edit a run in the "Run Definition" window. This allows the user to copy a previous run and then change the component to be analyzed.

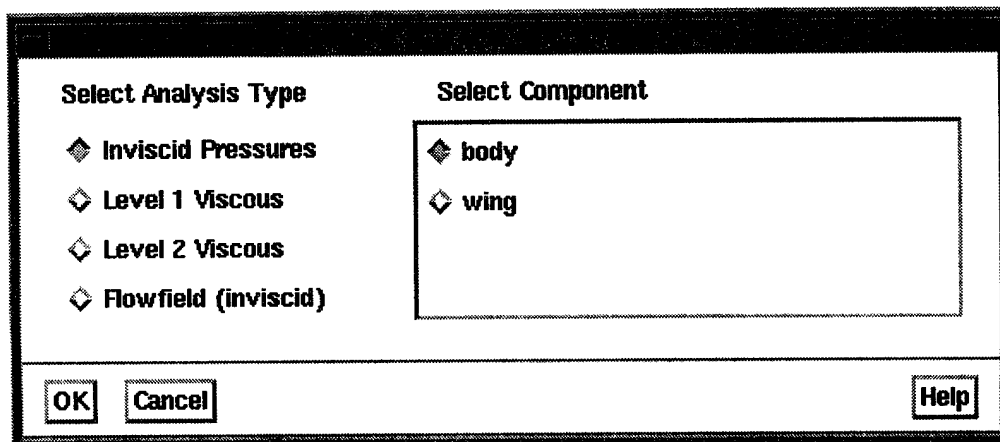


Figure 4.2-2 Analysis Type/Component

### 4.3 Flowfield/Shock Shape Analysis

This section discusses the input required to analyze shocks and the resultant flowfield within the shocklayer. An overview of the flowfield/shock shape methods is also presented.

The capability to calculate the flowfield/shock shape has been improved in S/HABP Mark V. The flowfield methods implemented into S/HABP Mark V include the second order shock expansion method and a blunt body starting solution. The capability to calculate the shock shape and flow properties along streamlines generated by QUADSTREAM has been added to the existing cutting plane option. The capability to calculate the flowfield properties throughout the shock layer at 12 points from the body to the shock has been added. Several improvements to new and existing methods such as the flowfield interpolation routine, tracking of multiple shocks and the cutting plane option were made. The capability to write out the shock shape and flowfield properties to a VUAERO results file to enable graphically displaying the shock shape and flowfield now exist in S/HABP. In addition, the flowfield data can be stored for later use in an inviscid analysis and/or written to the output file.

#### 4.3.1 General Inputs

The flowfield inputs have changed substantially from the Mark IV version of S/HABP. The most notable changes are related to the way flowfield properties are input. This is due to the implementation of new interpolation methods for both surface data properties and flowfield properties which eliminate the concept of subregions and the need to define special orientation axes. Another notable change is that the Mach number, angle-of-attack and sideslip angle are taken from the case definition and are no longer an input to the flowfield routines. The "Flowfield Inputs" window is shown in Figure 4.3-1 and is used to access all flowfield/shock shape input. The window is laid out such that the user chooses the type of analysis and only those inputs that correspond to the chosen analysis are made active. The window is divided into three parts. The top section of the window contains the "Component:" field which identifies the current component; the "Flowfield Analysis Type" input; access to the "Shock Expansion" inputs, "Hand Input Surface Data" inputs and "Non-Uniform Flowfield Data" inputs; and additional output options. The bottom center section of the window contains the inputs for the "Simple Flowfield Methods." And the bottom section of the window contains the "Uniform Flowfield" inputs.

Component: unnamed

Flowfield Analysis Type: **Flowfield- Shock Expansion** ☐

Edit Shock Expansion Inputs...  
 Edit Hand Input Surface Data...  
 Edit Non-Uniform Flowfield Data...

Write Detailed Output: **No** ☐

---

**Shock Output**

Write Vuaero Results File: **No** ☐

Number of Shock Points per Streamline **50**

Spacing of Shock Points Along Streamline: **Evenly Distributed** ☐

---

Simple Flowfield Method: **Wedge Compression** ☐

Freestream Mach	<b>5.00</b>	Vx/V	<b>-1.000000</b>	Alpha-Beta 1 <input type="checkbox"/>
Flow Turning Angle	<b>0.00</b>	Vy/V	<b>0.000000</b>	<b>Apply Flowfield</b>
		Vz/V	<b>0.000000</b>	

---

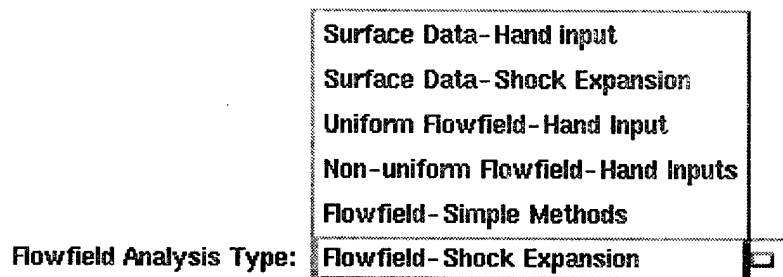
**Uniform Flowfield Data**

Mach	<b>5.00</b>	Vx/V	<b>-1.00</b>	Alpha-Beta 1 <input type="checkbox"/>
P/P inf	<b>0.00</b>	Vy/V	<b>0.00</b>	<b>Apply Flowfield</b>
T/T inf	<b>0.00</b>	Vz/V	<b>0.00</b>	

**OK** **Apply** **Cancel** **Help**

**Figure 4.3-1 Flowfield Analysis Window**

The "Flowfield Analysis Type" option menu is shown in Figure 4.3-2 and is used to select the type of analysis for the current component. There are six types of analysis that can be specified from hand input to shock expansion as shown in Figure 4.3-2. Each analysis type has its own set of inputs that must be entered. To aid the user to enter only the required inputs, all unnecessary inputs are grayed out. The inputs for "Flowfield - Simple Methods" and "Uniform Flowfield - Hand Input" options are contained on the "Flowfield Inputs" main window. All other inputs are accessed through additional dialog windows as described below.



**Figure 4.3-2 Flowfield Analysis Type**

The required inputs for the more detailed analysis types are accessed through the “Edit” buttons shown in Figure 4.3-3. The inputs for the “Surface Data - Hand Input” option are accessed through the “Edit Hand Input Surface Data ...” button. The inputs for “Surface Data - Shock Expansion” and “Flowfield - Shock Expansion” options are accessed through the “Edit Shock Expansion Inputs ...” button. The inputs for “Non-Uniform Flowfield - Hand Inputs” option are accessed through the “Edit Non-Uniform Flowfield Data ...” button. These buttons launch additional windows to input the required data.

**Edit Shock Expansion Inputs...**  
**Edit Hand Input Surface Data...**  
**Edit Non-Uniform Flowfield Data...**

**Figure 4.3-3 Access To Flowfield Analysis Inputs**

The detailed output option is shown in Figure 4.3-4 and is used to write detailed flowfield or surface data to the standard output file.

**Write Detailed Output:** ☐ **No** ☐

**Figure 4.3-4 Detailed Flowfield Output Option**

The VUAERO Results File contains a description of the shock shape and the flowfield data for the current component. The file can be used to graphically view the shock shape and flowfield using the VUAERO code. The inputs shown in Figure 4.3-5 are used to write the VUAERO results file. The user clicks the “Write VUAERO Results File” button to write the file. The user can customize the shock output by specifying the number of points and the spacing of the points along the shock. Two options are given to space the points; (1) Even distribution, (2) Half Cosine distribution. The half cosine distribution gives a denser distribution towards the front and is recommended for blunt nose vehicles.

### Shock Output

Write Vuaero Results File: ☐ No ☐

Number of Shock Points per Streamline

Spacing of Shock Points Along Streamline: ☐ Evenly Distributed ☐  
☐ Half Cosine Spacing

Figure 4.3-5 VUAERO Results File Inputs

### 4.3.2 Shock Expansion Inputs

The shock expansion inputs have not changed significantly from the Mark IV version of S/HABP with the exception of adding inputs to use a blunt body starting solution and read streamlines. The "Shock Expansion Inputs" window is shown in Figure 4.3-6 and is used to enter all inputs required to analyze a vehicle using shock expansion. The window is divided into two main sections. The left section contains general inputs and the right side contains inputs specifically for generating cutting planes.

General Inputs	Cutting Planes
Shock Expansion Order: <input type="checkbox"/> Second Order <input type="checkbox"/>	Number of cutting planes: <input type="text" value="7"/>
Flowline Source: <input type="checkbox"/> Generate Cutting Planes <input type="checkbox"/>	Cutting plane type: <input type="checkbox"/> Meridian Cuts <input type="checkbox"/>
Starting Solution: <input type="checkbox"/> Tangent Cone ARC CP792 <input type="checkbox"/>	Cutting plane spacing: <input type="checkbox"/> Evenly Spaced <input type="checkbox"/>
Body Slope: <input type="checkbox"/> Linear Slopes <input type="checkbox"/>	Surface normal: <input type="checkbox"/> Use Orientation <input type="checkbox"/>
Additional Output: <input type="checkbox"/> None <input type="checkbox"/>	Detail Print Flag: <input type="checkbox"/> Do not print <input type="checkbox"/>
Axial length of blunted nose: <input type="text" value="1.000"/>	<b>Cutting Plane Axis System</b>
Body radius aft of bluntness: <input type="text" value="1.000"/>	X <input type="text" value="0.000"/> PSI <input type="text" value="0.00"/>
	Y <input type="text" value="0.000"/> THETA <input type="text" value="0.00"/>
	Z <input type="text" value="0.000"/> PHI <input type="text" value="0.00"/>
	<b>Meridian Planes</b>
	Phi (deg) <input type="text"/>
	<input type="text"/>
	<b>Parallel Planes</b>
	Plane angle (deg): <input type="text"/>
	X Y Z
	<input type="text"/>

Figure 4.3-6 Shock Expansion Input Window

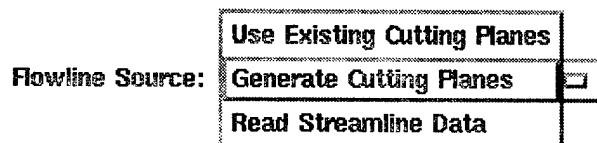
**4.3.2.1 General Shock Expansion Input** - This section discusses the general inputs required to analyze a vehicle using shock expansion.

The order of the shock expansion analysis can be specified using the “Shock Expansion Order” option menu shown in Figure 4.3-7. The user can choose either first order or second order shock expansion. The default input is for second order shock expansion.



**Figure 4.3-7 Shock Expansion Order**

The shock expansion flowfield is calculated along flowlines. The flowlines can be defined using cutting planes or streamlines. The user should reference Section 4.3.7 Limitations for additional information about using streamlines for flowfield analysis. The “Flowline Source” option menu is shown in Figure 4.3-8 and is used to choose the type and source of the flowlines. The “Use Existing Cutting Planes” option can be used on consecutive flowfield runs for the same component since the cutting plane data already exist. The “Generate Cutting Planes” is the default option and is used to generate cutting planes. When this option is used the user must input the cutting plane data on the right side of the “Shock Expansion Input” window. The “Read Streamline Data” option is used to read streamline data from an existing file. When using the streamline option, a streamline file must exist with the same alpha-beta combinations defined for the corresponding case.



**Figure 4.3-8 Flowline Source Option Menu**

Figure 4.3-9 shows the available starting solutions for the shock expansion calculations. In addition to the baseline tangent wedge and tangent cone, a blunt nose starting solution has been added. The blunt nose starting solution is applicable to both two dimensional (cylinders) and three dimensional (spheres) blunted bodies. If either of the blunt nose starting solutions are used, the user must enter the nose length and body diameter as shown in Figure 4.3-12.



Starting Solution:

Tangent Wedge (2-D)
Tangent Cone ARC CP792 <input type="checkbox"/>
Tangent Cone (Jone's)
Blunt Nose (Spherical)
Blunt Nose (2-D Circular Cyl)

Figure 4.3-9 Shock Expansion Starting Solution Menu

The "Body Slope" option menu is shown in Figure 4.3-10 and is used to enter the type of slopes used in the shock expansion calculations. The two options are linear slopes and circular arc slopes. The default option is linear slopes.

Body Slope:

Linear Slopes <input type="checkbox"/>
Circular Arc

Figure 4.3-10 Body Slope Option Menu

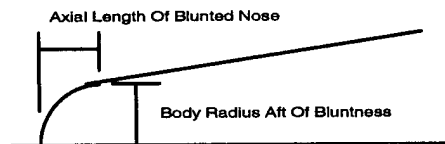
The "Additional Output" option menu is shown in Figure 4.3-11 and is used to print detailed shock expansion data to the standard output file.

Additional Output:

None <input type="checkbox"/>
Print Detailed SOSE Data

Figure 4.3-11 Shock Expansion Output Options

Figure 4.3-12 shows the inputs for the blunt nose starting solution. The blunt nose starting solution requires that the distance aft of the nose tip to the end of the bluntness be entered in addition to the radius of the body at that point. This information is used to calculate an equivalent spherical nose radius.

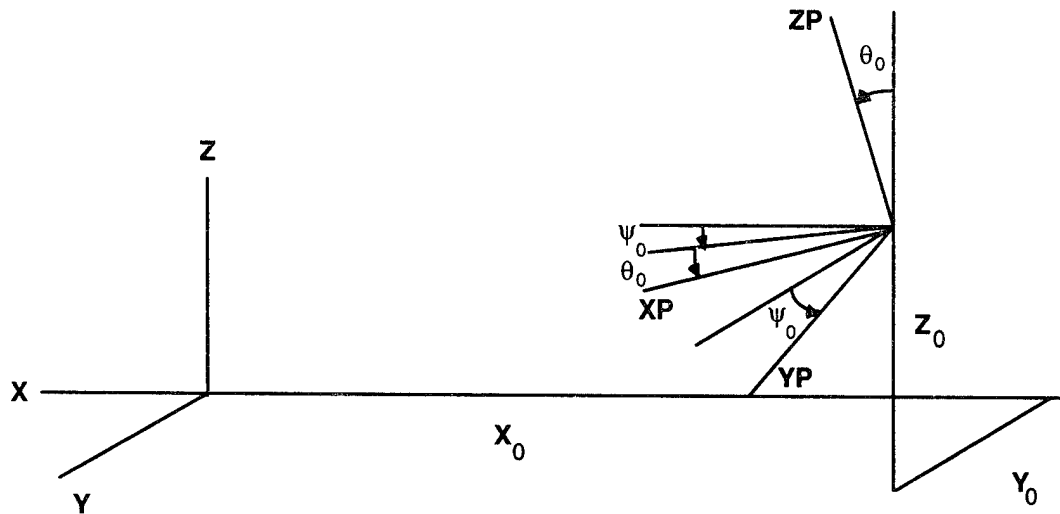


Axial length of blunted nose:

Body radius aft of bluntness:

Figure 4.3-12 Blunt Nose Starting Inputs

**4.3.2.2 Cutting Plane Inputs** - Two classes of cutting planes may be selected: meridian and parallel cuts. The cutting planes are defined with respect to an axis whose orientation may be arbitrarily specified in the reference geometry (body) coordinates. The cutting plane axis is initially assumed to be coincident with the reference axis system and is translated by  $x_0$ ,  $y_0$ ,  $z_0$  and rotated by  $\psi_0$ ,  $\theta_0$  as shown in Figure 4.3-13. An initial meridian angle  $\phi_0$  may also be input. All cutting planes will be perpendicular to the YP, ZP plane. Parallel cutting planes will be parallel to the XP axis and meridian planes will contain the XP axis.



**Figure 4.3-13 Cutting Plane Axis Orientation To Reference Axis System**

**Meridian Planes:** Meridian planes are specified by a rotation,  $\phi$ , about the XP axis, where  $\phi = 0$  is the negative ZP plane as shown in Figure 4.3-14. Meridian planes may be specified individually ( $\phi$  input in ascending order) or selected as equally spaced. Equally spaced meridian planes are calculated using the following equation.

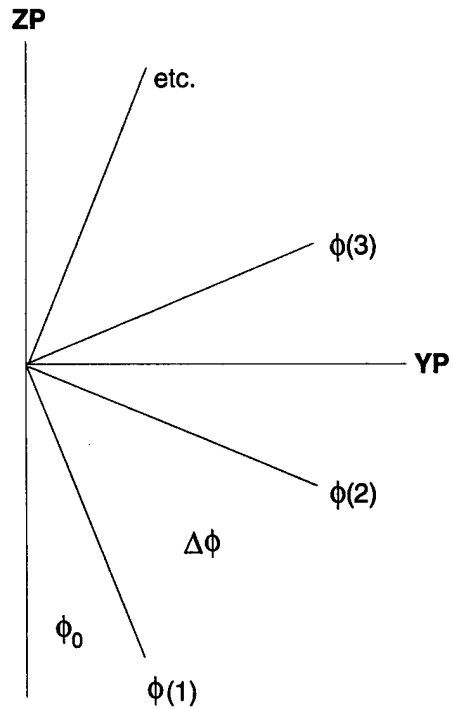
$$\phi(I) = \phi_0 + (I-1) \times \Delta\phi ; I = 1, NPL$$

where,

NPL = Number of cutting planes

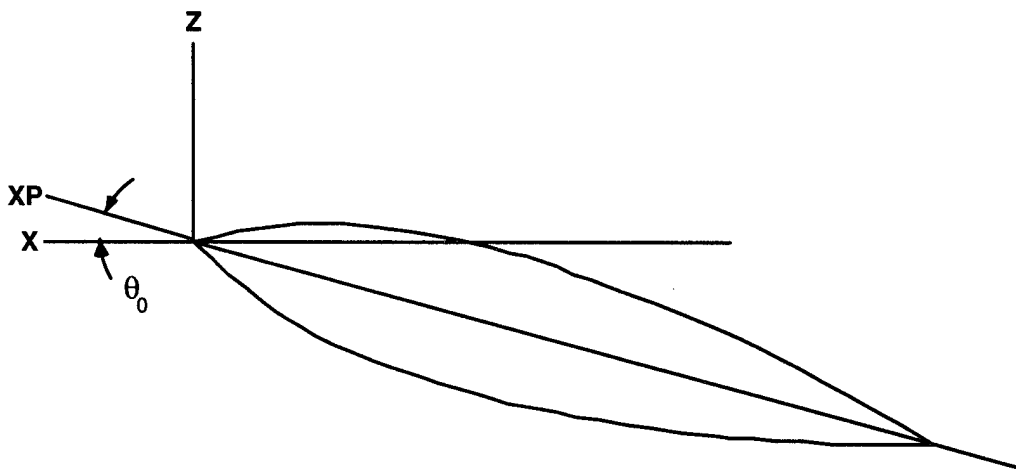
$$\Delta\phi = 180 / (NPL-1)$$

**Note:** Meridian Planes are “one-sided” such that intersections with the surface are found only at  $\phi$  and not at  $\phi + 180$ .



**Figure 4.3-14 Meridian Plane Orientation**

Some care must be taken in defining the cutting plane axis in relation to the surface geometry. Consider the example of an inclined fuselage as shown in Figure 4.3-15. If the axis for the meridian planes is specified coincident with the X-body axis ( $\psi_0 = 0$  and  $\theta_0 = 0$ ), meaningless results will be obtained for the flowlines aft of the location at which the body is below the X-axis. Correct orientation of the XP-axis in this case would be at a negative  $\theta_0$ , passing through the nose and the trailing edge points. In general, the meridian axis should be fully within the component being analyzed.



**Figure 4.3-15 Meridian Plane Example**

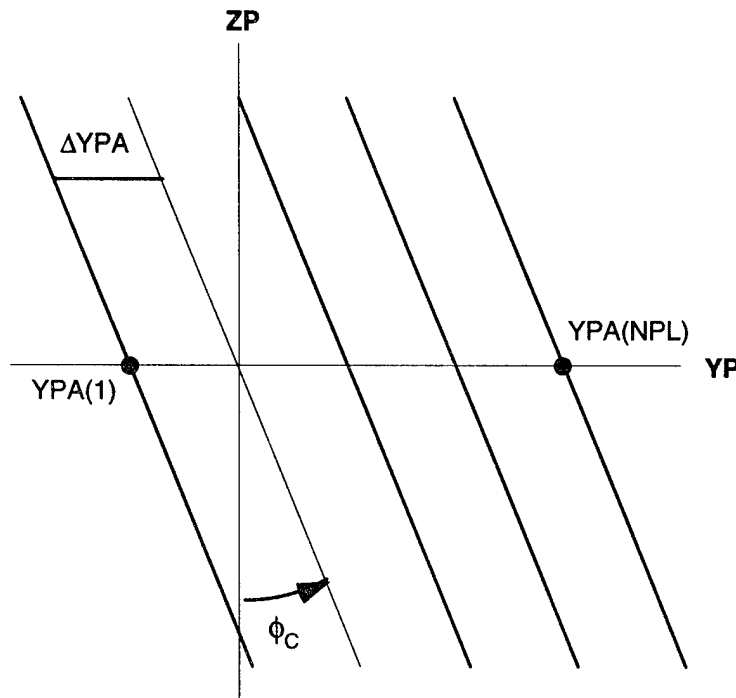
**Parallel Planes:** Parallel cutting planes are inclined to the ZP-axis at a constant angle  $\phi_c$ . Parallel planes can be input individually or using the equally spaced option. The input position are entered in reference (Body) coordinates and are automatically transformed to the cutting plane coordinate system. Equally spaced parallel planes are calculated using the following equation.

$$YPA(I) = YPA(1) + [(I-1) \times \Delta YPA] ; I = 1, NPL$$

where,

NPL = Number of cutting planes

$$\Delta YPA = [YPA(NPL) - YPA(1)] / (NPL-1)$$



**Figure 4.3-16 Parallel Cutting Planes**

**Note:** Parallel cutting planes are “two-sided” in that intersections with the surface are found at both  $\phi_c$  and  $\phi_c + 180$ . Wing like components should therefore be defined in separate groups of upper and lower panels.

**Cutting Plane Inputs:** The following inputs are used to define cutting planes to be used with the shock expansion flowfield analysis.

The “Number of Cutting Planes” input field is shown in Figure 4.3-17 and is used to enter the number of cutting planes. The default number is seven cutting planes with a maximum of 35 cutting planes allowed by S/HABP.

Number of cutting planes:

**Figure 4.3-17 Number Of Cutting Planes**

The “Cutting Plane Type” option menu is shown in Figure 4.3-18 and is used to specify the type of cutting planes to be used for the flowfield analysis. The default option is “Meridian Cuts” which is used for body type components. The “Parallel Cuts” option is used to distribute cutting planes in parallel planes. This option is used for wing type components.

Cutting plane type: 

Meridian Cuts	<input checked="" type="checkbox"/>
Parallel Cuts	<input type="checkbox"/>

**Figure 4.3-18 Cutting Plane Type Option Menu**

The “Cutting Plane Spacing” option menu is shown in Figure 4.3-19 and is used to specify the spacing of the cutting planes. The “Evenly Spaced” option is used to evenly distribute the cutting planes either in meridian cuts or parallel cuts. If the “Evenly Spaced” option is used with meridian cuts, the cutting planes are evenly distributed from  $\text{PHI} = 0.0$  to  $\text{PHI} = 180.0$ . If the “Evenly Spaced” option is used with parallel planes, the user must specify the beginning plane and ending plane to distribute the cutting planes. The beginning and ending planes are specified using the “Parallel Plane Spacing” input field as described later in this section. The “Input Spacing” option is used to distribute the cutting planes at various points specified by the user. When the “Input Spacing” option is chosen, the user must enter the number of cutting planes specified in the “Number of Cutting Planes” field. If the “Input Spacing” option is used with meridian cuts, the user must enter the PHI angles for all the cutting planes in the “Meridian Plane Spacing” input field as defined in a subsequent paragraph. If the “Input Spacing” option is used with parallel cuts, the user must specify each plane using the “Parallel Plane Spacing” input field as discussed below.

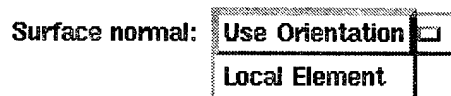
Cutting plane spacing: 

Evenly Spaced	<input checked="" type="checkbox"/>
Input Spacing	<input type="checkbox"/>

**Figure 4.3-19 Cutting Plane Spacing Option Menu**

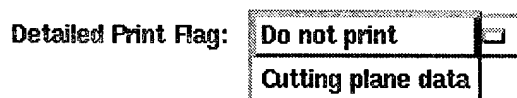
The shock and resultant flowfield are calculated in a plane along the flowline. The plane in which the flowfield is calculated can be described by the cutting plane or by the local element

normal. The user should reference Section 4.3.7 for additional information about using the local element normal for the flowfield analysis. The “Surface Normal” option menu is shown in Figure 4.3-20 and is used to specify the plane orientation to be used in the flowfield analysis. The “Use Orientation” option specifies to use the cutting plane orientation for the flowfield analysis. The “Local Element” option specifies to use the local element normal as the plane orientation for the flowfield analysis.



**Figure 4.3-20 Cutting Plane Surface Normal Option Menu**

The “Detailed Print Flag” option menu is shown in Figure 4.3-21 and is used to write detailed cutting plane data out to the standard output file.



**Figure 4.3-21 Cutting Plane Detailed Print Option**

The “Cutting Plane Axis System” input fields are shown in Figure 4.3-22 and are used to specify the orientation of the cutting planes. The X, Y and Z inputs are used to specify the origin of the cutting plane axis and are input in geometry units. PSI, THETA and PHI are used to specify the angles relative to the body axis and are input in degrees.

Cutting Plane Axis System			
X	<input type="text" value="0.000"/>	PSI	<input type="text" value="0.00"/>
Y	<input type="text" value="0.000"/>	THETA	<input type="text" value="0.00"/>
Z	<input type="text" value="0.000"/>	PHI	<input type="text" value="0.00"/>

**Figure 4.3-22 Cutting Plane Orientation Input**

The “Meridian Planes” input field is shown in Figure 4.3-23 and is used to input the spacing of meridian planes when the “Cutting Plane Type” is set to “Meridian” and the “Cutting Plane Spacing” option is set to “Input Spacing.” The user enters angles relative to the plane of symmetry starting at the bottom centerline. The number of angles should be the same as the number specified in the “Number of Cutting Planes” input field.

Each angle is entered on a separate line. Additional data can be entered anywhere in the list. Data can be deleted by deleting the angle on the line desired. Note: The blank line will remain

until either the “Apply” button or the “OK” button is clicked. Modifications can be made to the inputs by moving the pointer to the line displaying the angle desired and editing the value.

**Figure 4.3-23 Input Meridian Cutting Plane Spacing**

The “Parallel Planes” input field is shown in Figure 4.3-24 and is used to input the spacing of parallel planes when the “Cutting Plane Type” is set to “Parallel” and the “Cutting Plane Spacing” option is set to either “Input Spacing” or “Evenly Spaced.” If the “Cutting Plane Spacing” option is set to “Evenly Spaced,” only the beginning and ending cutting planes need to be entered. If the “Cutting Plane Spacing” option is set to “Input Spacing,” then the number of planes should be the same as the number specified in the “Number of Cutting Planes” input field. The “Plane Angle” input field is used to enter the constant offset to the Z cutting plane axis. The X,Y,Z input field is used to enter a point that the cutting plane passes through.

Each set of coordinates is entered on a separate line separated by a space or comma. Additional data can be entered anywhere in the list. Data can be deleted by deleting the values on the line desired. Note: The blank line will remain until either the “Apply” button or the “OK” button is clicked. Modifications can be made to the inputs by moving the pointer to the line displaying the data desired and editing the values.

X	Y	Z
0.000	0.000	0.000
0.000	0.000	0.000
0.000	0.000	0.000
0.000	0.000	0.000
0.000	0.000	0.000

**Figure 4.3-24 Input Parallel Cutting Plane Spacing**

### 4.3.3 Surface Data Inputs

The Hand Load Surface Data inputs have been modified to ease the entry of experimental data. Pressure data can now be loaded in any order without concern about subregions. In addition, unnecessary inputs have been eliminated. The "Hand Input Surface Data" window is shown in Figure 4.3-25 and is used to enter surface pressure coefficients at points on the geometry.

Each surface pressure coefficient and its corresponding body coordinates are entered on a separate line separated by a space or comma. The X, Y and Z coordinates are entered first followed by the pressure coefficient. Additional data can be entered anywhere in the list. Data can be deleted by deleting the values on the line desired. Note: The blank line will remain until either the "Apply Flowfield" button or the "Apply" button is clicked. Modifications can be made to the inputs by moving the pointer to the line displaying the data desired and editing the values.

The flowfield data must be input for all flight conditions specified for the corresponding case. For example, if 10 alpha, beta combinations have been defined for the case which the flowfield data will be used, then the user must enter 10 sets of data. The data displayed for editing is for the Alpha-Beta set shown. The user clicks on the arrows next to the Alpha-Beta Set to change the active set. The user must click the "Apply Flowfield" button to save the flowfield inputs for the current Alpha-Beta set before entering data for the next set.

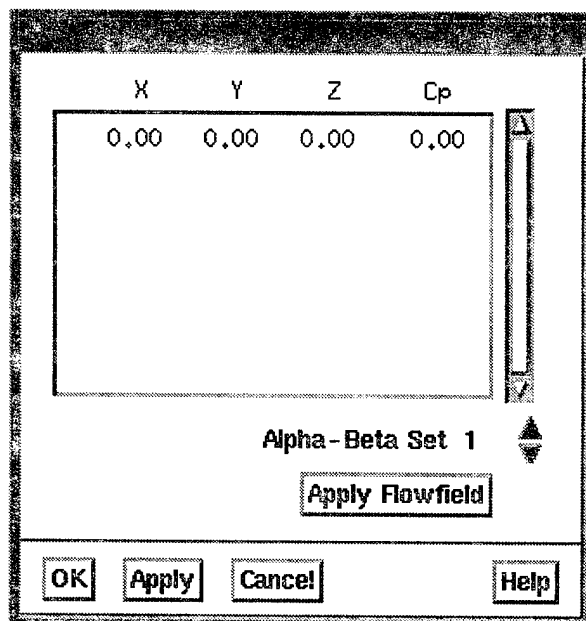


Figure 4.3-25 Hand Input Surface Data Window



#### 4.3.4 Uniform Flowfield Inputs

The "Uniform Flowfield Data" inputs are shown in Figure 4.3-26 and are used to enter uniform flowfield data on the Flowfield storage unit for use in the inviscid pressure analysis. The inputs consist of the Mach number, three components of the velocity vector, pressure ratio and temperature ratio.

The flowfield data must be input for all flight conditions specified for the corresponding case. For example, if 10 alpha, beta combinations have been defined for the case which the flowfield data will be used, then the user must enter 10 sets of data. The data displayed for editing is for the Alpha-Beta set shown. The user clicks on the arrows next to the Alpha-Beta Set to change the active set. The user must click the "Apply Flowfield" button to save the flowfield inputs for the current Alpha-Beta set before entering data for the next set.


Uniform Flowfield Data			
Mach	<input type="text" value="5.36"/>	Vx/V	<input type="text" value="-1.00"/>
P/P inf	<input type="text" value="0.00"/>	Vy/V	<input type="text" value="0.00"/>
T/T inf	<input type="text" value="0.00"/>	Vz/V	<input type="text" value="0.00"/>
			Alpha - Beta Set 1 
			<input type="button" value="Apply Flowfield"/>

Figure 4.3-26 Uniform Flowfield Inputs

#### 4.3.5 Non-Uniform Flowfield Inputs

The Non-Uniform Flowfield Data cards have been modified to ease the entry of experimental data. Non-Uniform flowfield data can now be loaded in any order without concern about subregions. The "Non-Uniform Flowfield Inputs" window is shown in Figure 4.3-27 and is used to enter non-uniform flowfield data on the Flowfield storage unit for use in the inviscid pressure analysis.

Each flowfield data point must be entered on a separate line. Each entry on the line must be separated by a space or comma. The X, Y and Z coordinates are entered first, followed by the Mach number, three components of the local flow, pressure ratio and temperature ratio. Additional data can be entered anywhere in the list. Data can be deleted by deleting the values on the line desired. Note: The blank line will remain until either the "Apply Flowfield" button or the "Apply" button is clicked. Modifications can be made to the inputs by moving the pointer to the line displaying the data desired and editing the values.

The flowfield data must be input for all flight conditions specified for the corresponding case. For example, if 10 alpha, beta combinations have been defined for the case which the flowfield data will be used, then the user must enter 10 sets of data. The data displayed for editing is for

the Alpha-Beta set shown. The user clicks on the arrows next to the Alpha-Beta Set to change the active set. The user must click the “Apply Flowfield” button to save the flowfield inputs for the current Alpha-Beta set before entering data for the next set.

X	Y	Z	Mach	Vx	Vy	Vz	P/Pinf	T/Tinf
0.0	0.0	0.0	0.00	0.0000	0.0000	0.0000	0.0000	0.0000

Alpha-Beta Set 1

Apply Flowfield

OK Apply Cancel Help

Figure 4.3-27 Non-Uniform Flowfield Data Window

#### 4.3.6 Simple Flowfield Inputs

The “Simple Flowfield Methods” inputs are shown in Figure 4.3-28 and are used to enter uniform flowfield data on the Flowfield storage unit for use in the inviscid pressure analysis. Four different methods are available to generate the flowfield. The inputs for all methods are the freestream Mach number, three components of the velocity vector, and the flow turning angle.

The flowfield data must be input for all flight conditions specified for the corresponding case. For example, if 10 alpha, beta combinations have been defined for the case which the flowfield data will be used, then the user must enter 10 sets of data. The data displayed for editing is for the Alpha-Beta set shown. The user clicks on the arrows next to the Alpha-Beta Set to change the active set. The user must click the “Apply Flowfield” button to save the flowfield inputs for the current Alpha-Beta set before entering data for the next set.

Simple Flowfield Method:		<div> <div>Wedge Compression</div> <div>Prandtl-Meyer Expansion</div> <div>Cone Surface Flowfield</div> <div>Newtonian Prandtl-Meyer</div> </div>		<input type="checkbox"/>	
Freestream Mach	<input type="text" value="5.00"/>	Vx/V	<input type="text" value="-1.000000"/>	Alpha-Beta Set	1
Flow Turning Angle	<input type="text" value="0.00"/>	Vy/V	<input type="text" value="0.000000"/>	<input type="button" value="Apply Flowfield"/>	
		Vz/V	<input type="text" value="0.000000"/>		

Figure 4.3-28 Simple Flowfield Inputs

#### 4.3.7 Limitations

The flowfield methods implemented in S/HABP allow the user to either calculate or input flowfield data to be used in the inviscid pressure routine. The shock expansion method is the principal method to calculate the shock and resultant flowfield. It is also the most complex and has some limitations which are discussed in this section. The user should reference the VECC Final Report for additional information.

The capability to calculate the flowfield along streamlines has been implemented in addition to the cutting plane option that already existed in S/HABP. The streamline option provides a quick method to define the flowlines necessary to perform the shock expansion calculation. However, streamlines at angle-of-attack do not produce realistic shock shapes. The problem is that the shock shape is calculated along the streamline, normal to the surface. This is the same problem with using the local element normal for cutting planes. S/HABP uses the local orientation to map the 2-D shock calculated in the shock expansion routines into the 3-D body coordinates. The shock shape produced using the surface normal is more technically correct from a pure theoretical viewpoint; however the shock using the cutting plane orientation is a better approximation of the actual shock. It is recommended that cutting planes be used for most configurations. Streamlines can be used, but it is recommended that only streamlines at zero degrees angle of attack be used.

The blunt body starting solution implemented into S/HABP Mark V is used in conjunction with the second order shock expansion (SOSE) to provide the capability to analyze vehicles with blunt noses. The blunt body starting solution calculates the shock standoff distance, shock bluntness and shock radius of curvature. These values are then used to calculate the shock shape and resulting shock angle in which the flowfield properties can be calculated. The shape of the detached shock is assumed to be a portion of a hyperbola, parabola, prolate ellipse, sphere or an oblate ellipse. The blunt body starting solution implemented into S/HABP Mark V is valid for

spherically blunted noses for Mach numbers between 1.2 and 10.0 and zero degrees angle-of-attack. However, this method has been extended by calculating an equivalent body slope aft of the bluntness, which will account for angle-of-attack. The equivalent body slope aft of the spherical nose must be less than 40 degrees. In addition to the above limitations, the blunt nose region flowfield data is only calculated at the body and directly behind the shock.

During implementation of the algorithm to write out the VUAERO results file, several issues arose that limited the usefulness of the shock output. First, the current implementation of the shock shape calculation produces a shock boundary even in regions where there is only expansion. This is not an error, but a way to represent the bounds between the expanded flow and freestream flow, which is necessary to perform the flowfield interpolation. Another significant problem is caused by embedded shocks. The original implementation wrote out the shock using the shortest shock length; however, this caused the primary bow shock to terminate prematurely when an embedded shock was present. This problem was attenuated when an intersection occurred on one shock line and not the remaining shock lines. The solution implemented is to write out all shock points for all shocks calculated. This allows the user to see all the shocks and any points of intersection. However, it does not produce a single 3-D bow shock which accounts for secondary shocks. Another problem associated with embedded shocks is that VUAERO connects all shock points for each shock. That is, all primary shock points are connected, all secondary shock points are connected and so forth. The problem arises when a flowline on the bottom produces a second shock and the flowline on the top produces a second shock but no intermediate flowlines produce secondary shocks. In this case VUAERO still connects the bottom and top secondary shock points which produces a poor representation of the actual shock. Even though, the shock is not represented correctly in VUAERO, the flowfield data is handled correctly in S/HABP.

#### **4.3.8 Output**

The data generated by the flowfield routines can be saved for later use by the inviscid routines or can be written to the standard output file and/or to a VUAERO results file. All output generated by the flowfield routines is optional output such that print flags must be set to obtain the desired output. Print flags are available to write out cutting plane data, detailed SOSE data, and the flowfield properties between the surface and shock for each point along the flowline. It should be noted that the last option will produce a large amount of data. In addition to the tabular data available for output, a print flag can be set to generate a VUAERO results file that contains the shock definition and the flowfield properties between the surface and shock.

The tabular data that are written out to the standard output file has been modified to account for the addition of the new methods implemented in S/HABP Mark V. The biggest modification is the addition of the flowfield properties throughout the shock layer for each point along the

flowline. This output consist of the x,y,z coordinates, pressure ratio, density ratio, temperature ratio, local Mach number and the direction cosines of the local flow. Example tabular output for cutting plane data, detailed shock expansion data, and the flowfield properties are shown in Figures 4.3-29, 4.3-30 and 4.3-31 respectively for a simple cone-cylinder.

The capability to write the shock shape and flowfield data to a VUAERO results file is now included in S/HABP. Logic has been added to write out both the shock shape and flowfield properties to a VUAERO results file to enable graphically displaying the shock shape and flowfield in VUAERO. The VUAERO results file allows viewing of the shock geometry, pressure ratio, density ratio, temperature ratio and the direction cosines of the local flow. For a full description of the VUAERO results file see the VUAERO Reference manual(Reference 2).

```

      CONFIGURATION          NUMBER OF CUTS = 5          TOTAL POINTS = 25
PHIO = 0.0000    PSIO = 0.0000    THEO = 0.0000    XO = 0.0000    YO = 0.0000    ZO = 0.0000

PLANE NUMBER 1          PHI = 0.000000    YPA = 0.000000    NUMBER OF INTERSECTIONS = 5

POINTS ORDERED FOR A AS PER IOG = -1 AND IOR(I) = 0 0 0 0 0 0 0 0 0 0
PANEL NAME      J          X          Y          Z          R          A
Ellipse         1          0.000000    0.000000    0.000000    0.000000    0.000000
Ellipse         1         -10.000000    0.000000    -2.500000    2.500000   -10.000000
Ellipse        14         -20.000000    0.000000    -5.000000    5.000000   -20.000000
Ellipse        27         -50.000000    0.000000    -5.000000    5.000000   -50.000000
Ellipse        40         -80.000000    0.000000    -5.000000    5.000000   -80.000000
    0 DUPLICATE POINTS REMOVED FROM THE DATA.

      .
      .
      .

PLANE NUMBER 5          PHI = 180.000000    YPA = 0.000000    NUMBER OF INTERSECTIONS = 5

POINTS ORDERED FOR A AS PER IOG = -1 AND IOR(I) = 0 0 0 0 0 0 0 0 0 0
PANEL NAME      J          X          Y          Z          R          A
Ellipse        13          0.000000    0.000000    0.000000    0.000000    0.000000
Ellipse        13         -10.000000    0.000000    2.500000    2.500000   -10.000000
Ellipse        26         -20.000000    0.000000    5.000000    5.000000   -20.000000
Ellipse        39         -50.000000    0.000000    5.000000    5.000000   -50.000000
Ellipse        52         -80.000000    0.000000    5.000000    5.000000   -80.000000
    0 DUPLICATE POINTS REMOVED FROM THE DATA.

```

**Figure 4.3-29 Detailed Cutting Plane Output**

# SECOND-ORDER SHOCK-EXPANSION

M =	5.000000	ALFWD =	0.000000				
A	R	DELTA	S	PHIT	AMP	ALFP	
0.0000	0.0000	14.0362	0.0000	0.0000	5.0000	0.0000	
-10.0000	2.5000	14.0362	10.3078	0.0000	5.0000	0.0000	
-20.0000	5.0000	14.0362	20.6155	0.0000	5.0000	0.0000	
-50.0000	5.0000	0.0000	50.6155	0.0000	5.0000	0.0000	
-80.0000	5.0000	0.0000	80.6155	0.0000	5.0000	0.0000	

SOSE STARTING FLOW - M = 4.02450 THETA = 19.139 DPSK = 0.43340

INVISCID SOLUTION, SURFACE

A	P/P(INF)	M	CP	PB	ETA	ANGLE	CPLIM	PC
-5.0000	3.40209	4.0245	0.1373	3.40209	0.00000	28.4237	0.1373	3.40209
1ST ORDER SHOCK EXPANSION								
-15.0000	3.40209	4.0245	0.1373	3.40209	-0.54506	28.4237	0.1373	3.40209
-35.0000	0.76339	5.2427	-0.0135	0.66562	0.69172	11.1608	0.0000	1.00000
-65.0000	0.88153	5.1164	-0.0068	0.83257	0.69172	11.3463	0.0000	1.00000

INVISCID SOLUTION, SHOCK WAVE 1

AS	RS	P2/P1	M2	THETA
0.0000	0.0000	2.9687	4.0245	19.1395
-14.9980	5.2051	2.9687	4.0245	19.1394
-29.9961	10.4102	2.9687	4.0245	11.5370

Figure 4.3-30 Detailed Shock Expansion Output

FLOW FIELD DATA FOR STREAMLINE ROW 1  
INTERPOLATED POINT NUMBER 1

X	Y	Z	LOCAL MACH	* FLOW DIRECTION COSINES *			P/PINF	T/TINF	DENSITY RATIO
				VX	VY	VZ			
-10.000	0.000	-2.500	4.02450	-0.97014	0.00000	-0.24254	3.40209	1.41532	2.27742
10.000	0.000	-2.588	4.02023	-0.97048	0.01121	-0.23013	3.39927	1.41709	2.27631
10.000	0.000	-2.676	4.01627	-0.97083	0.02361	-0.21639	3.39525	1.41872	2.27471
10.000	0.000	-2.765	4.01255	-0.97122	0.03685	-0.20166	3.39019	1.42026	2.27271
10.000	0.000	-2.853	4.00905	-0.97162	0.05073	-0.18617	3.38415	1.42170	2.27031
10.000	0.000	-2.941	4.00575	-0.97205	0.06510	-0.17007	3.37714	1.42306	2.26752
10.000	0.000	-3.029	4.00264	-0.97250	0.07982	-0.15348	3.36918	1.42435	2.26435
10.000	0.000	-3.118	3.99972	-0.97298	0.09478	-0.13652	3.36024	1.42555	2.26078
10.000	0.000	-3.206	3.99698	-0.97348	0.10986	-0.11930	3.35032	1.42669	2.25681
10.000	0.000	-3.294	3.99443	-0.97401	0.12497	-0.10191	3.33937	1.42774	2.25242
10.000	0.000	-3.382	3.99206	-0.97457	0.13999	-0.08446	3.32735	1.42872	2.24759
10.000	0.000	-3.471	3.98987	-0.97516	0.15480	-0.06706	3.31421	1.42962	2.24230

INTERPOLATED POINT NUMBER 2

X	Y	Z	LOCAL MACH	* FLOW DIRECTION COSINES *			P/PINF	T/TINF	DENSITY RATIO
				VX	VY	VZ			
-20.000	0.000	-5.000	4.02450	-0.97014	0.00000	-0.24254	3.40209	1.41532	2.27742
20.000	0.000	-5.176	4.02444	-0.97184	0.00712	-0.22737	3.35454	1.41534	2.25850
20.000	0.000	-5.353	4.02711	-0.97326	0.03969	-0.18780	3.31053	1.41425	2.24082
20.000	0.000	-5.529	4.02884	-0.97423	0.08705	-0.13571	3.28076	1.41354	2.22876
20.000	0.000	-5.706	4.02861	-0.97478	0.14229	-0.07787	3.26566	1.41363	2.22261
20.000	0.000	-5.882	4.02633	-0.97510	0.19630	-0.02252	3.26004	1.41457	2.22032
20.000	0.000	-6.059	4.02231	-0.97559	0.23180	0.01496	3.25231	1.41622	2.21716
20.000	0.000	-6.235	4.01683	-0.97634	0.24614	0.03238	3.23952	1.41847	2.21192
20.000	0.000	-6.412	4.01006	-0.97714	0.25109	0.04049	3.22758	1.42126	2.20702
20.000	0.000	-6.588	4.00225	-0.97791	0.25065	0.04303	3.21770	1.42446	2.20296
20.000	0.000	-6.765	3.99367	-0.97864	0.24721	0.04232	3.21010	1.42799	2.19982
20.000	0.000	-6.941	3.98463	-0.97933	0.24270	0.04025	3.20465	1.43170	2.19757

Figure 4.3-31 Detailed Flowfield Output

## 4.4 Inviscid Analysis

The most widely used portion of S/HABP is the inviscid pressure analysis which is used to determine the inviscid forces and moments of the configuration. This section discusses the input required for the S/HABP inviscid pressure analysis whose input window is shown in Figure 4.4-1. This window is divided into three window panes. The left most window pane contains input fields for the basic pressure methods. The top right window panes contains input used for additional analysis such as shielding and flowfield interference. The text fields in this window pane that are followed by three dots "..." are actually text buttons which bring up windows for further input. Input required for options selected in the top right corner of the window pane are discussed further in Sections 4.4.4, 4.4.5, and 4.4.6. The lower right portion of the inviscid pressure methods window contains buttons for additional output which is discussed in Section 4.4.7.

**Component:** Trapezoidal Body

Windward Method:

Leeward Method:

Dynamic Pressure Ratio:

Not used:

Not used:

Surface Slope Modification Factor:

Impact Shock Method:

Component Type:

Control Deflection:  deg.

Hinge Line:

	Root	Tip
X	<input type="text"/>	<input type="text"/>
Y	<input type="text"/>	<input type="text"/>
Z	<input type="text"/>	<input type="text"/>

☐ Apply Shielding Analysis on this Component  
Edit Shielding Inputs for this Component...

☐ Use Previously Calculated Flowfield Data To Calculate Interference On this Component  
Edit Flowfield Interference Inputs...

☐ Use Stored Input Pressures for this Component  
Edit Input Pressures for this Component...

**Detail Print Flags**

☒ Print Detailed Force Contributions for Each Element

☒ Print Detailed Local Property Calculations and Iteration Results

☐ Print VUAERO surface pressure file

OK Apply Cancel Help

Figure 4.4-1 Inviscid Pressure Input Window

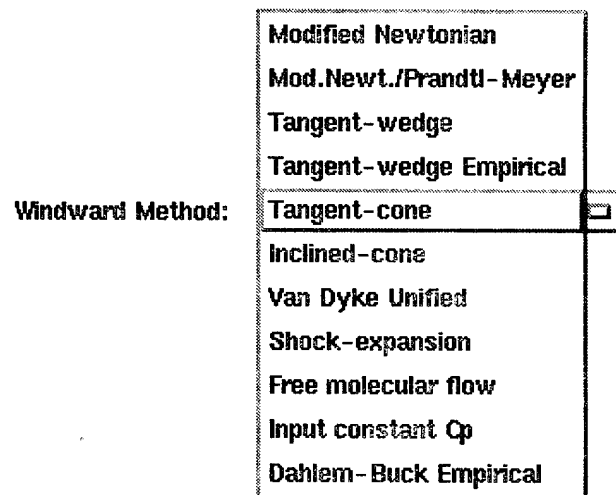
#### 4.4.1 Pressure Method Selection

The first input to be made is the pressure method for both the windward and leeward side of the component. Access to the pressure methods is via an option menu next to the labels "Windward Method" and "Leeward Method." Immediately below the pressure methods are four text fields. The input required (and label shown) depends on which pressure methods are selected. Each one of the pressure methods, and any additional input required, are defined in the following two sections.

**4.4.1.1 Windward Methods** - There are 11 windward pressure methods available as shown in Figure 4.4-2. Additional input, if required for the selected method is also described below. The user must specify the ratio of local to freestream dynamic pressure (default value 1.0) and the surface slope modification factor regardless of the pressure method selected.

The surface slope modification factor ENPM is usually input as zero. If it is not zero, the surface slope,  $\theta$ , will be divided by this number. The impact angle is then calculated as:

$$\delta = \pi / 2 - \theta_{\text{input}} / \text{ENPM}$$



**Figure 4.4-2 Windward Inviscid Pressure Methods**

Modified Newtonian	Use Modified Newtonian Equation, $C_p = K \sin^2 \delta$ The user must input the Newtonian correction factor K. In pure Newtonian flow ( $M=\infty$ , $\gamma=1$ ), $K=2$ . (Default, $K=1.83$ )
--------------------	---



Mod. Newt/Prandtl-Meyer	Modified Newtonian theory is used along the body until a point is reached where both the pressure and the pressure gradients match those that would be calculated by a continuing Prandtl-Meyer Expansion. The user must input the Newtonian correction factor, K, and the Prandtl-Meyer expansion correction factor, $\eta_c$ (usually input as 1).
Tangent Wedge	Tangent-Wedge method uses the oblique shock relationships of NACA TR-1135 (Reference 7).
Tangent Wedge Empirical	Empirically derived wedge impact angle relationship.
Tangent-Cone	An approximate solution for predicting surface flow on a cone in supersonic flow for attached bow shock conditions.
Inclined Cone	A method for predicting pressures on circular cones at angle-of-attack.
Van Dyke Unified	Based on the unified supersonic-hypersonic small disturbance theory proposed by Van Dyke. Useful for thin profile shapes down to the supersonic speed regime.
Shock-Expansion	Based on classical shock-expansion theory. Surface elements are handled in a "strip-theory" manner. The user must also specify the method for determining the characteristics of the first element of a column via the option menu labeled "Impact Shock Method" which becomes active when this method is selected.
Free-Molecular Flow	Used for rarefied flow regime where the mean free path of the molecules is much greater than the characteristic body dimension. User must also input the free molecular flow parameter, $f_n$ , the ratio of the body temperature to the freestream temperature $T_B / T_\infty$ , and the tangential momentum accommodation coefficient, $f_t$ . The normal momentum accommodation coefficient, $f_n$ , and the tangential momentum accommodation coefficient, $f_t$ , equal zero for Newtonian flow and equal 1.0 for completely diffuse reflection.
Input Constant Cp	The user supplies a constant pressure coefficient to be applied to all elements of this component. Pressure coefficient values input in the dialogue area below the pressure method selection menus.
Dahlem-Buck Empirical	Method developed by Dahlem-Buck. This is an empirical method which approximates tangent cone pressure at low impact angles and approaches Newtonian values at large impact angles.

**4.4.1.2 Leeward Methods** - There are 12 leeward pressure methods available as shown in Figure 4.4-3. Additional input if required for the selected method is described below.

Leeward Method:

Newtonian
Mod.Newt./Prandtl-Meyer
Prandtl-Meyer
Inclined-cone
Van Dyke Unified
High Mach Base
Shock-expansion
Input constant Cp
Free molecular flow
Dahlem-Buck mirror
ACM Empirical
Half Prandtl-Meyer

**Figure 4.4-3 Leeward Inviscid Pressure Methods**

Newtonian	$C_p = 0$
Mod. New./Prandtl-Meyer	Modified Newtonian theory is used along the body until a point is reached where both the pressure and the pressure gradients match those that would be calculated by a continuing Prandtl-Meyer Expansion. The user must input the Newtonian correction factor, K, and the Prandtl-Meyer expansion correction factor, $\eta_c$ (usually input as 1).
Prandtl-Meyer	Use Prandtl-Meyer expansion from freestream.
Inclined Cone	A method for predicting pressures on circular cones at angle-of-attack.
Van Dyke Unified	Based on the unified supersonic-hypersonic small disturbance theory proposed by Van Dyke. Useful for thin profile shapes down to the supersonic speed regime.
High Mach Base	$C_p = -1/M_\infty^2$
Shock-Expansion	Based on classical shock-expansion theory. Surface elements are handled in a "strip-theory" manner. The user must also specify the method for determining the characteristics of the first element of a

column via the option menu labeled "Impact Shock Method" which becomes active when this method is selected.

Input Constant Cp	The user supplies a constant pressure coefficient to be applied to all elements of this component. Pressure coefficient value is input in the dialogue area below the pressure method selection menus.
Free-Molecular Flow	Used for rarefied flow regime where the mean free path of the molecules is much greater than the characteristic body dimension. User must also input the free molecular flow parameter, $f_n$ , the ratio of the body temperature to the freestream temperature $T_B / T_\infty$ , and the tangential momentum accommodation coefficient, $f_t$ . The normal momentum accommodation coefficient, $f_n$ , and the tangential momentum accommodation coefficient, $f_t$ , equal zero for Newtonian flow and equal 1.0 for completely diffuse reflection.
Dahlem-Buck Mirror	Method developed by Dahlem-Buck. This is an empirical method.
ACM Empirical	Empirical method derived from the Aerodynamic Configured Missile program. Useful for body shapes in the supersonic/ low hypersonic regime.
Half Prandtl-Meyer	One half of the pressure obtained using Prandtl-Meyer expansion from freestream. Based on high Mach number test data results.

#### 4.4.2 Component Type

The option menu labeled "Component Type," shown in Figure 4.4-4, allows the user to define whether a component is a fixed panel, pitch control surface, or a yaw/roll control surface. A pitch control surface such as a flap, or elevator is deflected symmetrically. A yaw/roll control surface such as an aileron is deflected differentially. The terms symmetric and differential used here relate to whether the mirrored control surface deflects in the same direction or the opposite direction as the input control surface (remember that only the left side of the vehicle is defined). For example, deflection of a trailing flap is usually a symmetric control deflection since both the left and right control surfaces deflect in the same direction (i.e., if the left trailing edge is deflected downward then the right flap is mirrored and also deflects downward). In general, rudder and aileron deflections are considered differential deflection. The aileron case is easy to visualize; when one trailing edge deflects downward the other deflects upward. A rudder deflection of twin vertical tails is also a differential deflection to S/HABP. In this case when the left rudder is deflected trailing edge toward the centerline, the right rudder is deflected trailing edge away from the centerline to generate yaw control. Thus the right rudder is not the mirror image of the left rudder and is therefore considered a differential deflection.

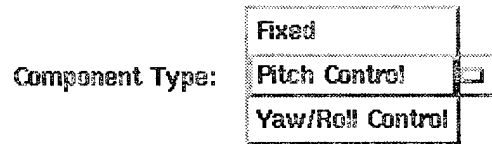


Figure 4.4-4 Component Type Option Menu

#### 4.4.3 Control Deflection/Hinge Line

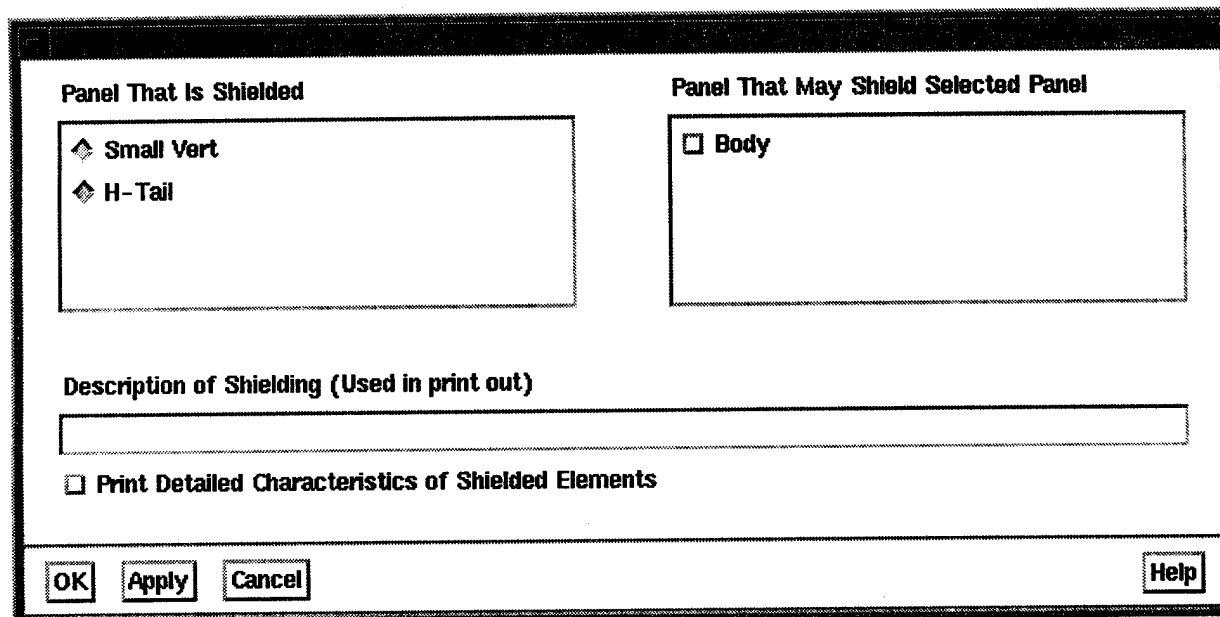
The control deflection and hinge line dialogue boxes become active if the component is designated a control surface. The deflection angle is input for the left control surface with a positive deflection being determined by the right hand rule. Place your right thumb pointing from tip to root and your fingers will wrap in the direction of a positive deflection. For example consider a left horizontal surface, pointing your right thumb from tip to root, your fingers wrap in the leading edge up direction (or trailing edge down depending on your point of view). Thus a positive deflection for a horizontal surface is trailing edge down. A positive deflection of a vertical tail is trailing edge left when looking from the rear. A positive aileron deflection places the left trailing edge down and the right trailing edge up thus generating a positive rolling moment.

The hinge line defines the point about which the control deflections occur. The user defines the hinge line by providing the X,Y,Z coordinates of the hinge line at the root and at the tip of the control surface. Remember that X-axis is positive forward.

#### 4.4.4 Apply Shielding Analysis

The user may perform a shielding analysis on any panel to account for the situation where one part of a vehicle shape is shielded from the freestream flow by another part of the vehicle. The shielding program will search for elements which shield a portion of the panel of interest. The portion of the panel which is shielded will be stored as quadrilateral elements with negative surface area. Thus as the element forces are summed for the component in question the force on the shielded area has a net sum of zero and will have no contribution to the final component forces.

This option is turned on via the "Inviscid Pressures" window by pushing in the "Apply Shielding Analysis On This Component" button. To launch the "Shielding Analysis" input window shown in Figure 4.4-5, the user clicks on the "Edit Shielding Input For This Component ..." text button.



The dialog box is titled "Shielding Analysis Input Window". It contains two main sections at the top. The left section, titled "Panel That Is Shielded", contains a list with two items: "Small Vert" and "H-Tail", each preceded by a diamond-shaped radio button. The right section, titled "Panel That May Shield Selected Panel", contains a single item: "Body", preceded by a square checkbox. Below these sections is a text area labeled "Description of Shielding (Used in print out)". At the bottom of the dialog, there is a checkbox labeled "Print Detailed Characteristics of Shielded Elements". The bottom of the window features four buttons: "OK", "Apply", "Cancel", and "Help".

**Figure 4.4-5 Shielding Analysis Input Window**

The user must define which panels are to be considered as shielded and which panels may be shielding them. After launching the shielding input window, all of the panels of the current component are listed in the left dialogue area called "Panel That Is Shielded" and each panel has a radio button before the name. The user can perform a shielding analysis on each panel, but must select one panel at a time (via the radio button) when selecting which panels may be shielded. Panels that may be shielding the selected panel are listed on the right of the window and are selected via the buttons before their names.

The natural tendency is to select every panel as potentially shielding the panel of interest. The user is cautioned to not over specify the potential shielding panels. This can lead to the same portion of the panel being shielded by multiple panels and thus double account for the shielded area.

The user can print out details of the quadrilateral elements that have been calculated in the shielding analysis by pressing the button labeled "Print Detailed Characteristics Of Shielded Elements" located at the bottom of the window. Detailed data is written to the standard output file. In addition, a VUAERO neutral file which contains the geometry of the negative area elements is created. This file is named SHIELD.DAT.N (see Figure 1.1-2).

#### **4.4.5 Apply Flowfield Interference**

Flowfield interference is used to apply local flowfield properties in the inviscid pressure calculations. The inputs consist of defining the local flowfield and the radius of influence used in the interpolation. The radius of influence is used to control the number of points used in the

interpolation. The interpolation method uses a distance weighted average to determine the flow properties at a point within the flowfield by inversely weighting each point within the radius of influence by its distance from the point of interest. If no data are found within the specified radius the local conditions at the desired point would default to freestream. This is a fast and efficient method to determine the flowfield properties at any point in the flowfield. The interpolated values are relatively insensitive to the number of flowfield points and radius of influence as long as the desired point is contained within the volume defined by the flowfield.

The "Flowfield Interference" window is shown in Figure 4.4-6 and is used to access flowfield data for interference calculations. The component to which the interference will be applied is listed in the upper left corner of the window. The user can select three different sources to obtain the flowfield data for use in the interference calculations. The "Hand input uniform flowfield data" option is used to enter a uniform flowfield via inputs at the bottom of the window. The "Uniform data from flowfield unit" option is used to apply an uniform flowfield that has previously been loaded on the flowfield mass storage unit. The "Non-uniform data from flowfield unit" option is used to apply non-uniform data or shock expansion flowfield data previously loaded on the flowfield mass storage unit. The user can specify to write the interpolated flowfield values to the standard output file by selecting the "Print interpolated data" option under the "Optional Output" menu. By default, the interpolated data are not printed to the standard output file. If a non-uniform flowfield is used for interference, the user must enter a radius of influence for the interpolation routine.

**Component** Wing

**Interference Source** Non-uniform data from flowfield unit

**Previously Defined Flowfield Runs**

body, Flowfield, off, N/A

**Optional Output** None

**Radius of influence:** 1.000000

**Uniform Flowfield Data**

Mach 5.00 Vx/V -1.00 Alpha-Beta Set 1

P/P inf 0.00 Vy/V 0.00 Apply Flowfield

T/T inf 0.00 Vz/V 0.00

OK Apply Cancel Help

Figure 4.4-6 Flowfield Interference Window

**4.4.5.1 Hand Input Uniform Flowfield Data** - The “Uniform Flowfield Data” inputs are shown at the bottom of Figure 4.4-6 and are used to enter hand input uniform flowfield data for flowfield interference calculations. The inputs consist of the Mach number, three components of the velocity vector, pressure ratio and temperature ratio.

The flowfield data must be input for all flight conditions specified in the corresponding case. For example, if 10 alpha, beta combinations have been defined in the case, the user must enter 10 sets of data. The data displayed in the bottom window pane is for the Alpha-Beta set shown. The user clicks on the arrows next to the Alpha-Beta Set to change the active set. The user must click the “Apply Flowfield” button to save the flowfield inputs for the current Alpha-Beta set before entering data for the next set.

**4.4.5.2 Previously Calculated Flowfield Data** - This option allows the user to apply data from the flowfield routines for computing flowfield interference. Previously defined flowfield runs are selected using the “Previously Defined Flowfield Runs” selection box shown in Figure 4.4-6.

Remember only flowfield runs that have been previously defined are available in the selection box. The user selects the desired flowfield runs by clicking on the appropriate run or runs in the selection box. Up to five flowfield runs can be specified for use in the interference calculations.

#### **4.4.6 Use Stored Input Pressures**

The input pressure option has been modified to account for a new surface spline method implemented in S/HABP Mark V. The number of inputs have been substantially reduced. In addition, the inputs are more understandable. The only new input is the radius of influence which is used to control the number of points used in the surface spline. It is recommended that the radius of influence be set such that a minimum of eight points are available for interpolation.

The surface spline method implemented into S/HABP Mark V is a local interpolation scheme which determines the surface pressure at a given point based on pressures within a specified radius. When the surface spline method is used in conjunction with surface pressures calculated by SOSE, the search routine filters out all data points within the specified radius whose normals are greater than 90 degrees from the local element normal. If the surface spline routine is used in conjunction with hand loaded data, the user **MUST** enter the surface data on components in which all data points are valid for use in determining the surface pressure at the local point. (i.e., A wing would need to be broken into two components upper and lower, and the pressures loaded separately since pressures on the lower surface should not be used in the interpolation of pressures on the upper surface.). Also, a minimum of eight points are required to calculate the surface spline function. If less than eight points are available, the code defaults to a simple distance weighted average for the calculation of local surface pressure.

The "Input Pressure Option" window is shown in Figure 4.4-7 and is used to access surface pressure data for force and moment calculations. The component to which the pressures are applied is listed in the upper left corner of the window. The user can select from two sources of pressure data. The "Hand input Cp data stored on unit 10" option is used to apply surface pressure data that has previously been loaded on the flowfield mass storage unit. The "SOSE generated data stored on unit 10" option is used to apply pressure data calculated by the shock expansion routine and stored on the flowfield mass storage unit.

The user can specify to write the interpolated surface pressures to the standard output file by selecting the "Print interpolated data" option under the "Optional Output" menu. By default, the interpolated data are not printed to the standard output file. The user must enter a radius of influence for use in the interpolation process. The radius controls the amount data used in the interpolation.



Component: body

Flowfield Analysis Type: Hand input Cp data stored on unit 10

Previously Defined Flowfield Runs

body, Flowfield, off, N/A

Optional Output: None

Radius of Influence: 5.000000

OK Apply Cancel Help

Figure 4.4-7 Input Pressure Option Window

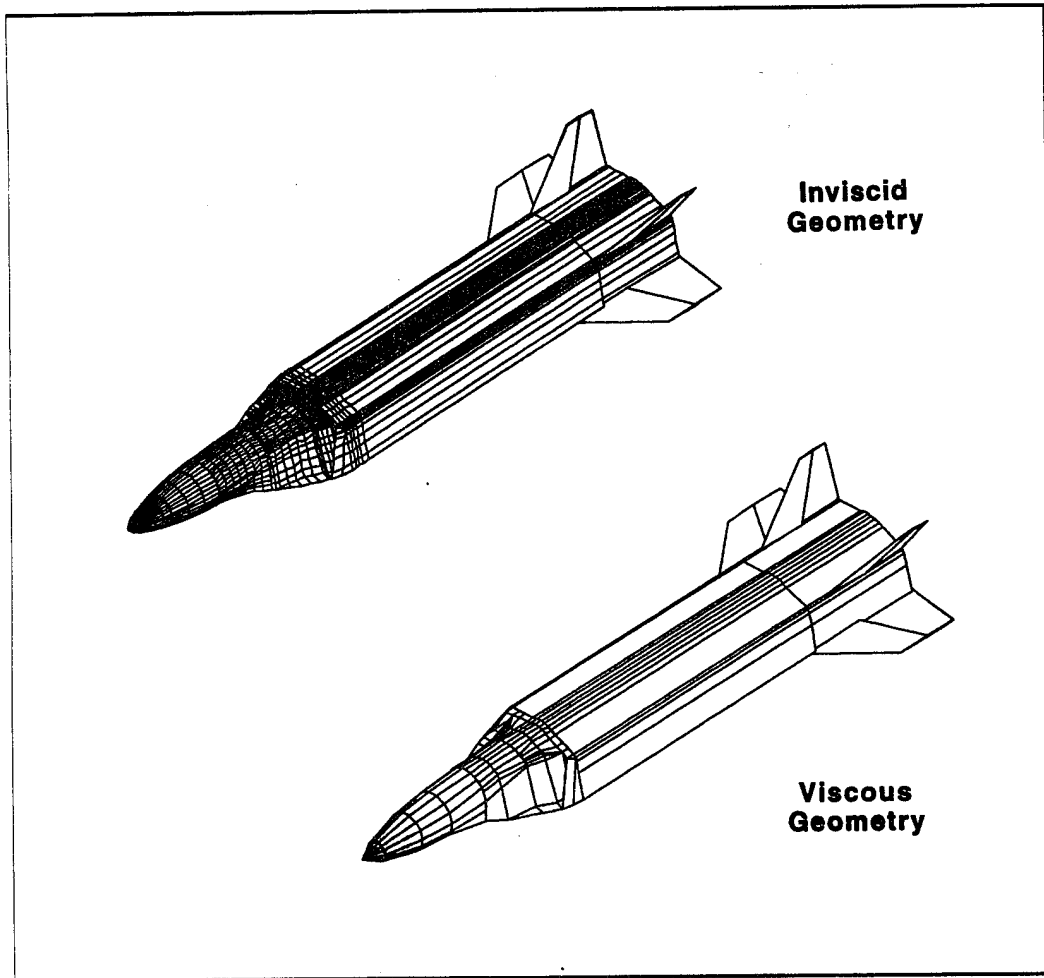
#### 4.4.7 Detailed Print Flags

The user can obtain additional output regarding the inviscid pressure analysis by pressing the buttons located in the lower right window pane of the "Inviscid Pressures" window (Figure 4.4-1). Two of the buttons allow more detail on the element properties however only one of them can be chosen at a time. They are the "Print Detailed Force Contributions for Each Element" button and "Print Detailed Local Property Calculations and Iteration Results" button. Both print options generate a significant amount of output. The third button located in this window pane allows the user to generate a pressure coefficient file which may then be displayed in the VUAERO program.

## 4.5 Level 1 Viscous Analysis

The Level 1 Viscous Analysis uses engineering methods first incorporated in the Mark III version of S/HABP to predict viscous forces and moments on a simplified geometry. No attempt is made to calculate the detailed skin friction distribution on the exact geometry shape, but rather, the vehicle is represented by a number of flat surfaces on each of which the shear force is determined. Reference temperature and reference enthalpy methods are available for both laminar and turbulent flows and, in addition, the Spalding-Chi method with either temperature or enthalpy ratios may be selected for turbulent calculations. The surface temperature may be either input or the radiation equilibrium value predicted by the code. The effect of planform shape, leading edge viscous-interaction, and the viscous force on blunt bodies are also considered.

The skin friction geometry is simplified to greatly reduce the amount of user defined input data. Generally leading edge surfaces and local curvature are omitted. Regions of relatively large curvatures can be represented by using a smaller number of elements. Figure 4.5-1 compares an inviscid (detailed) geometry with a viscous geometry. The viscous geometry can be easily created in the geometry modeler after the detailed model is complete. The user simply copies the detailed model panels and deletes unnecessary cross sections and grid points. The user is reminded to change the panel type to viscous by selecting "Panel Type" from the main window "Edit" menu (see Section 2.5 for more detail).



**Figure 4.5-1 Example Of Viscous Geometry Compared To Inviscid Geometry**

The "Level 1 Viscous Analysis" window is shown in Figure 4.5-2 and is comprised of four window panes. The left three window panes contain input fields for the viscous analysis. The right window pane contains input for the associated inviscid analysis required to generate pressure data for the viscous analysis. The right side of the window allows the user to set up both the viscous analysis and the required inviscid analysis in the same window rather than separately as in the Mark IV code (input to S/HABP actually remains the same, but the organization of the input was simplified). Each pane of the window will be discussed individually in the following sections.

<b>Component:</b> body		<b>Inviscid Analysis for Input to Viscous Routines</b>														
<b>Analysis Type:</b> Skin Friction <input type="checkbox"/>		<b>Windward Method:</b> Tangent-cone <input type="checkbox"/>														
<b>Viscous-Inviscid Interaction:</b> Tangent-Wedge <input type="checkbox"/>		<b>Leeward Method:</b> Van Dyke Unified <input type="checkbox"/>														
<b>Wall Temp. / Laminar / Turbulent</b> Calculate/Ref.Temp/Spalding-Chi <input type="checkbox"/>		<b>Dynamic Pressure Ratio:</b> 1.00 Not used: <input type="text"/> Not used: <input type="text"/>														
<b>Element Number</b> <table border="1"> <tr><td>Element 1</td></tr> <tr><td>Element 2</td></tr> <tr><td>Element 3</td></tr> <tr><td>Element 4</td></tr> <tr><td>Element 5</td></tr> <tr><td>Element 6</td></tr> <tr><td>Element 7</td></tr> <tr><td>Element 8</td></tr> <tr><td>Element 9</td></tr> <tr><td>Element 10</td></tr> <tr><td>Element 11</td></tr> <tr><td>Element 12</td></tr> <tr><td>Element 13</td></tr> </table>		Element 1	Element 2	Element 3	Element 4	Element 5	Element 6	Element 7	Element 8	Element 9	Element 10	Element 11	Element 12	Element 13	<b>Surface Element Data</b> Surface Element Area [sq-in]: <input type="text"/> Surface Element Length [in]: <input type="text"/> Surface Element Taper Ratio: <input type="text"/> Initial Surface Length [in]: <input type="text"/> Initial Surface Taper Ratio: <input type="text"/> Wall Temperature [deg R]: <input type="text"/> Flow Regime: Turbulent <input type="checkbox"/>	
Element 1																
Element 2																
Element 3																
Element 4																
Element 5																
Element 6																
Element 7																
Element 8																
Element 9																
Element 10																
Element 11																
Element 12																
Element 13																
Apply Skin Friction Input		<b>Surface Slope Modification Factor:</b> 1.0000 <b>Impact Shock Method:</b> Tangent-wedge <input type="checkbox"/>														
<b>Print Flags</b> <input type="checkbox"/> Detailed Skin Friction Intermediate Results <input type="checkbox"/> Skin Friction Data for Each Element <input checked="" type="checkbox"/> Print Iteration Results for Local Cf Data <input checked="" type="checkbox"/> Print Final Local Cf Data		<b>Component Type:</b> Fixed Surface <input type="checkbox"/> <b>Control Deflection:</b> <input type="text"/> deg. <b>Hinge Line:</b> <table border="1"> <thead> <tr> <th></th> <th>Root</th> <th>Tip</th> </tr> </thead> <tbody> <tr> <td>X</td> <td><input type="text"/></td> <td><input type="text"/></td> </tr> <tr> <td>Y</td> <td><input type="text"/></td> <td><input type="text"/></td> </tr> <tr> <td>Z</td> <td><input type="text"/></td> <td><input type="text"/></td> </tr> </tbody> </table>			Root	Tip	X	<input type="text"/>	<input type="text"/>	Y	<input type="text"/>	<input type="text"/>	Z	<input type="text"/>	<input type="text"/>	
	Root	Tip														
X	<input type="text"/>	<input type="text"/>														
Y	<input type="text"/>	<input type="text"/>														
Z	<input type="text"/>	<input type="text"/>														
OK Apply Cancel		Help														

Figure 4.5-2 Level 1 Viscous Analysis Window

#### 4.5.1 Level 1 Flags

The top left window pane shown in Figure 4.5-3 contains flags that are set for the viscous analysis of a component. The top of the window pane contains a label with the component name selected by the user before entering this window.

The user can select either "Skin Friction" analysis or "Induced Pressures" as the "Analysis Type" via an option menu. If the user selects "Induced Pressures," skin friction is not calculated rather induced pressures due to boundary layer displacement effects are calculated.

The user can select between "Tangent-Wedge" or "Tangent-Cone" for determining the "Viscous-Inviscid Interaction." This flag determines the type of pressure method used to correct the laminar skin friction and induced pressures for interaction between the inviscid pressure distribution and the boundary layer displacement.

<b>Component:</b>	body
<b>Analysis Type:</b>	<b>Skin Friction</b> <input type="checkbox"/>
<b>Viscous-Inviscid Interaction:</b>	<b>Tangent-Wedge</b> <input type="checkbox"/>
<b>Wall Temp. / Laminar / Turbulent</b>	
	<b>Calculate/Ref.Temp/Spalding-Chi</b> <input type="checkbox"/>

**Figure 4.5-3 Level 1 Viscous Flags**

The final option menu of this window pane allows selection of the wall temperature and skin friction methods. Figure 4.5-4 shows the options available. The basic wall temperature methods are: User Input, Adiabatic, and Calculated Equilibrium. The skin friction methods are selected in pairs. The first method listed in the method will be used if the flow is laminar. The second method listed is used when the flow is designated as turbulent. When wall temperature is to be input by the user, it is specified for each skin friction element in the left, middle window pane.

**Wall Temp. / Laminar / Turbulent**

**Calculate/Ref.Temp/Spalding-Chi**

**Adiabatic/Ref.Temp/Spalding-Chi**

**Input/Ref.Temp/Spalding-Chi**

**Calculate/Ref.Enthalpy/Spalding-Chi**

**Adiabatic/Ref.Enthalpy/Spalding-Chi**

**Input/Ref.Enthalpy/Spalding-Chi**

**Calculate/Ref.Temp/Ref.Temp**

**Input/Ref.Temp/Ref.Temp**

**Calculate/Ref.Enthalpy/Ref.Enthalpy**

**Input/Ref.Enthalpy/Ref.Enthalpy**

**Figure 4.5-4 Skin Friction Method Options**

#### **4.5.2 Skin Friction Element Data**

The left, middle window pane contains the bulk of the input for the level 1 viscous methods. On the left side of the window, are listed the element numbers for the component which are numbered sequentially beginning with the first panel of the component. On the right are the user input for each element (or a group of elements). Each of these inputs are further discussed below.

**IMPORTANT:** After selecting an element or group of elements, the user *must* click on the "Apply Skin Friction Input" button to save the input before selecting a new element (or elements) for input.

Element Number

Element 1  
Element 2  
Element 3  
Element 4  
Element 5  
Element 6  
Element 7  
Element 8  
Element 9  
Element 10  
Element 11  
Element 12  
Element 13

**Skin Friction Element Data**

Surface Element Area [sq-in]:

Surface Element Length [in]:

Surface Element Taper Ratio:

Initial Surface Length [in]:

Initial Surface Taper Ratio:

Wall Temperature [deg R]:

Flow Regime: ☒ Turbulent

**Apply Skin Friction Input**

**Figure 4.5-5 Skin Friction Element Input**

**4.5.2.1 Element Number** - Element numbering begins with the first panel of a component, and continues with panels in ascending order of panel number (i.e., as they are listed in the panel dialogue area of the main window). Skin friction components are often made up of single panels. To select an element for input, click on that element number, and it will be highlighted. To select a range of elements, click on the first element and then, holding the shift button, click on the last element. The entire range will be highlighted. To select several elements that are not in order, hold down the control button while clicking. Skin friction components may have a maximum of 100 elements.

Input for the skin friction elements is then made in the dialogue areas on the right side of this window pane. When several elements are selected, the values shown in the dialogue areas on the right are those of the first element selected.

**4.5.2.2 Surface Element Area** - The area of the element(s) selected is input here. The units match the input units defined via the main window "Edit" menu. If input as zero, S/HABP will calculate the surface area. The user may wish to define this input when a large element is approximating a curved surface such that the actual surface area is significantly different than the skin friction element area.

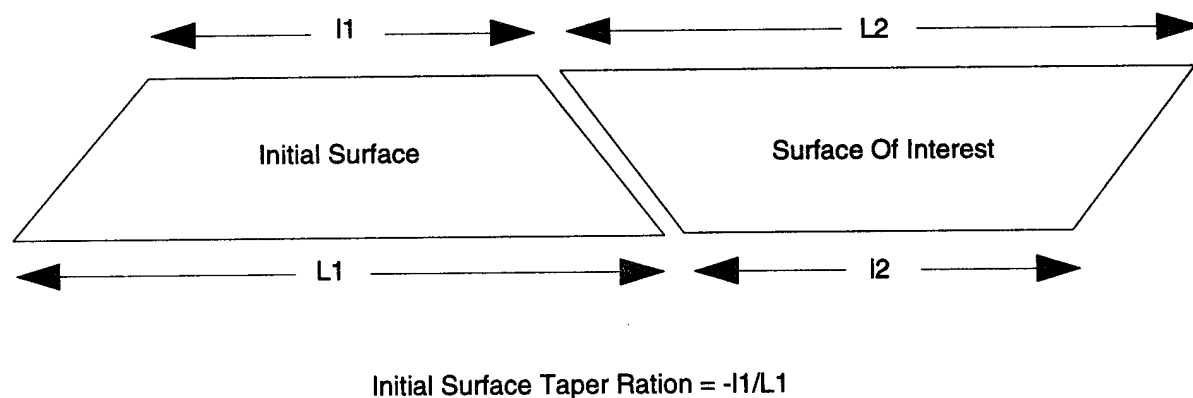
**4.5.2.3 Surface Element Length** - The length of the element(s) selected is input here. The units requested match the input units defined via the main window "Edit" menu, and are

converted to feet by the GUI when written to the S/HABP input file. If input as zero, S/HABP will calculate the length.

**4.5.2.4 Surface Element Taper Ratio** - The taper ratio is the length of the shortest side of the element divided by the longest side of the element. This input is always positive and 1.0 or less. If input as zero, S/HABP will calculate the taper ratio.

**4.5.2.5 Initial Surface Length** - The user should input the distance over which the flow would have traveled before reaching the element(s) selected. The units requested match the input units defined via the main window "Edit" menu, and are converted to feet by the GUI when written to the S/HABP input file.

**4.5.2.6 Initial Surface Taper Ratio** - The taper ratio of the initial surface is defined as the taper ratio of the shortest chord length to the longest chord length. If both the initial surface longest length and the longest length of the surface of interest are on the same edge of the shape, then the taper ratio of the initial surface is input as positive number. If these lengths are on opposite sides of the shape as shown in Figure 4.5-6, then the initial surface taper ratio is input as a negative number. The absolute value of the initial surface taper ratio is never greater than 1.0.



**Figure 4.5-6 Initial Surface Taper Ratio Example**

**4.5.2.7 Wall Temperature** - The wall temperature in degrees Rankine is input here if the user has selected user input wall temperature.

**4.5.2.8 Flow Regime** - The boundary layer state is specified via this option menu. The user can define whether each element is laminar or turbulent.

### 4.5.3 Print Flags

The bottom left window pane contains buttons which allow the user to select additional optional data to be written to the output file (.out). The print flag window pane is shown in Figure 4.5-7.

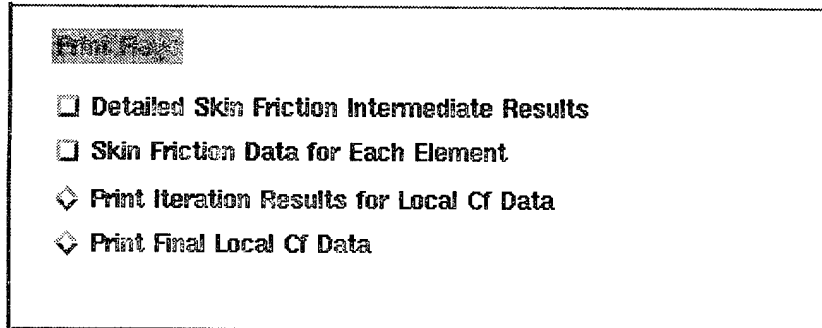


Figure 4.5-7 Skin Friction Optional Output

**4.5.3.1 Detailed Skin Friction Intermediate Results** - Print the detailed results of intermediate calculations for each element.

**4.5.3.2 Skin Friction Data For Each Element** - Print skin friction data for each surface element.

**4.5.3.3 Iteration or Final Local Cf Data** - The bottom two radio buttons allow the print out of local skin friction results. The user can select to print either iteration results for wall temperature and the final local skin friction data ("Print Iteration Result For Local Cf Data"), or only the final local skin friction data ("Print Final Local Cf Data").

### 4.5.4 Inviscid Analysis For Input To Viscous Routines

The right side of the window labeled "Inviscid Analysis For Input to Viscous Routines" is basically a repeat of the "Inviscid Analysis" window. Here the user defines the inviscid pressure methods which will be used to generate surface pressures required by the viscous analysis method. Refer to Section 4.4 for more detail on inviscid input.



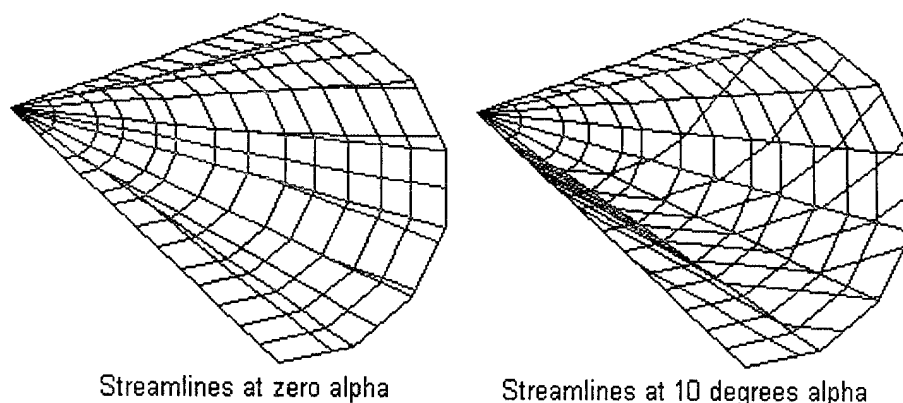
## 4.6 Level 2 Viscous Analysis

The new level 2 method enables the user to perform detailed viscous analysis on highly complex configurations that would be extremely time consuming with the level 1 method. Graphical input capability combined with automated data calculation relieves the user of tedious pre-analysis and manual input. In the new level 2 method, the user specifies inputs at the component level, not at the element level. The user no longer needs to calculate the initial length and the initial taper ratio for each element as required in level 1 because the flow calculation is based on streamlines rather than elements. Also the flow regime input for each element is no longer necessary because the new level 2 method includes flow transition and flow separation capabilities. Because of the reduced user input requirement, viscous analysis can now be performed on the more detailed inviscid geometry, eliminating the necessity of creating a simple viscous model.

The geometry must be broken into components in order to apply different methods to the different areas of the vehicle. For viscous analysis, the geometry is first divided into leading edge and flat plate areas. Leading edges are the front portion of the forebody, inlet, wings, fins, and tails which are more blunted than the rest of the structure. The remaining portion of the structure is modeled as flat plates. Maximum skin friction and heating usually occur at the leading edge areas. These areas should be modeled separately if they are a significant portion of the geometry or if the user is concerned about material temperature limits. However, for most cases, the leading edges are small compared to the total area, and only the analysis of the overall forces and moments is required. For these cases, the entire geometry is usually modeled as a flat plate and any inaccuracies in the leading edge portion is considered insignificant. The pressure method is the next most important basis for subdividing the geometry, especially for the flat plate components. Leading edge components can use the Modified Newtonian method. The flat plate areas often require different pressure methods such as tangent wedge for 2-D flow areas and tangent cone for axisymmetric flow areas. Depending on the level of detail required, the geometry can be further divided into smaller components based on any combination of the available methods.

The user must create streamlines before submitting the level 2 viscous S/HABP runs for analysis. The viscous calculation is performed along streamlines, and the results are then interpolated to the elemental centroids. Therefore, the user must ensure that the set of streamlines created has enough coverage over the components of interest. Figure 4.6-1 shows an example of inadequate streamline coverage caused by angle of attack. Both cones are covered with the same number of streamlines. The streamlines are evenly distributed over the left cone at zero alpha. At angle of attack, the streamlines converge towards the bottom centerline causing an uneven distribution as well as inadequate coverage on the top surface. In order to get

accurate interpolated results, the user must add more streamlines to the uncovered areas. A more detailed discussion of streamline coverage can be found in Section 3.2.2.



**Figure 4.6-1 Streamline Coverage Affects Accuracy of Level 2 Analysis**

In addition to streamline coverage, the user needs to check for premature stopping of streamlines. It is critical that streamlines for level 2 viscous analysis reach either a stagnation point or a stagnation line. Boundary layer and aeroheating calculations are highly dependent on the running length. Premature stopping of the streamlines would shorten the running length and cause an over-prediction of the results. Because streamlines are traced over components, the user must create components for all the areas between the components of interest and a stagnation point or line. In practice, it is often easiest for the user to create components over the entire geometry or at least on an entire part such as the body, wing, fin or tail which has its own local stagnation point or line. A more detailed discussion of premature stopping of streamlines can be found in Section 3.2.3.

The streamline file (.qstr) provides the level 2 viscous analysis with all the required geometric information. In addition, each streamline point is identified with a component ID which associates the streamline point with the appropriate set of boundary layer and aeroheating inputs. Streamline points on a component whose input is not specified uses the input specified for the previous point. If none of the previous points have been specified, the input is set to match that of the closest point aft on the streamline. The level 2 viscous input is described in the following section.

#### **4.6.1 Level 2 Viscous Input**

Level 2 viscous inputs are specified for each component in the *Level 2 Viscous Analysis* window shown in Figure 4.6-2. To access this window, choose *S/HABP Run Setup* from the main window edit menu which launches the *Run Definition* window where the user *Adds* a new run; on the *Analysis Type/Component* window, select the component and level 2 viscous analysis.

The component associated with the current set of inputs is identified at the top of the window. The remaining level 2 viscous inputs are described in the following sections.

Figure 4.6-2 Level 2 Viscous Analysis Window Specifies Input per Component

**4.6.1.1 Boundary Condition Input** - The boundary condition input, shown in Figure 4.6-3, include the *Wall Condition*, *Temperature*, *Emissivity* and *Radiation Sink Temperature*.

Figure 4.6-3 Boundary Condition Inputs

*Wall Condition* - The user may select either *Cold Wall* (constant wall temperature) or *Radiation equilibrium temperature* option from the wall condition option menu. *Cold Wall* is usually selected to simulate either a wind tunnel or flight test where the wall temperature data is available or to analyze a vehicle with the skin actively cooled to a known temperature. *Radiation equilibrium temperature* is used to analyze configurations with a thin metallic skin or a well insulated surface.

*Temperature* - If the cold wall option is selected, the user must specify the wall temperature (in Rankine) for the component.

*Emissivity* - The user must specified the surface emissivity if the radiation equilibrium temperature option is selected. Emissivity is a radiative property which indicates the percentage of the maximum thermal radiation emitted by the surface of a material. The value for emissivity ranges from 0 to 1, where one is an ideal radiator or *blackbody*. An emissivity of 0.8 is commonly used for aerospace composites and oxidized metals. . Metals such as aluminum and steel have emissivity as low as 0.02 to 0.3. These values are available in various handbooks on thermophysical properties of materials.

*Radiation sink temperature* - The user must also specify the radiation sink temperature (in Rankine) if the radiation equilibrium temperature option is selected. The sink temperature is the temperature of the surroundings with which the surface exchanges thermal radiation energy. The sink temperature of space is zero degree Rankine. For the earth's surface, the maximum ocean temperature of 528 degrees Rankine (86°F) is commonly use for the worst case estimate.

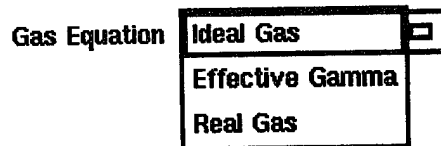
**4.6.1.2 Radius of Influence** - The radius of influence input box, shown in Figure 4.6-4, is located immediately below the radiation sink temperature input box.

Radius of Influence [in]:

**Figure 4.6-4 Input Box for the Radius of Influence Parameter**

The value is specified in the geometry units. The radius of influence controls the amount of data used for the interpolation from the streamlines to an element centroid. For each element, only the data from the streamline points which lie within the radius of influence are used for the interpolation. The interpolation requires that at least one streamline point fall within the radius of influence, but only the last 250 points read will be used. The actual value for the radius of influence is left to the user's judgment based on the component's size and streamline coverage.

**4.6.1.3 Gas Equation Flag** - The level 2 viscous method offers three gas models which are selected from the gas equation option menu shown in Figure 4.6-5.



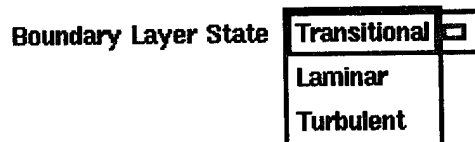
**Figure 4.6-5 Gas Models Available In Level 2 Viscous Analysis.**

*Ideal Gas* - The thermodynamic and transport properties are calculated with ideal gas equations. In the present implementation, air and helium are available. The gas is assumed to be thermally and calorically perfect. For the Level 2 analysis, Sutherland viscosity is used with a constant Prandtl number. The ideal gas calculation is the fastest and the most robust of the three gas models.

*Effective Gamma* - This method uses the ideal gas assumption with an effective ratio of specific heat ( $\gamma$ ) values of air. This value is obtained from a table lookup as a function of the flight altitude and freestream velocity. The effective gamma is a way to simulate real gas effects while maintaining the speed and robustness of the ideal gas model.

*Real Gas* - This option is selected to calculate the thermodynamic and transport properties with the NASA Ames equilibrium air model, RGAS. Transport properties are calculated using the UGAS model developed by Tannehill. Although equilibrium air calculations are more accurate than ideal gas for high speed flows, it is not nearly as fast nor as robust. Unlike the closed form ideal gas equations, RGAS and UGAS are curve fits which must be analyzed numerically. Equilibrium air calculation is usually 10 to 20 times slower than ideal gas and is more prone to errors because of its complexity and a more limited range of applicability.

**4.6.1.4 Boundary Layer State** - This menu allows the user to set the flow condition for the component as either Laminar, Turbulent or Transitional flow. These options are selected from the boundary layer state option menu shown in 4.6-6.

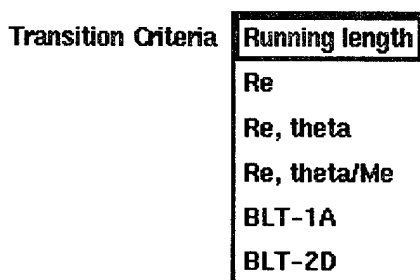


**Figure 4.6-6 Level 2 Viscous Analysis Includes Transition Capability**

In most applications, either the combination of laminar and turbulent flow or only transitional flow is recommended for a single configuration. In the first case the user knows the demarcation between laminar and turbulent areas and divides the configuration accordingly. Each component is then either all laminar or all turbulent. With the transitional flow option selected, the flow state (laminar, transitional and turbulent) is calculated along the streamlines. Once turbulence is

reached, all downstream points are set to turbulent even if a component whose flow properties meet the laminar flow criteria is encountered.

**4.6.1.5 Transition Criteria** - If the transitional boundary layer state option is chosen, the transition criteria option menu (Figure 4.6-7) becomes active. The user must select one of the six available transition criteria options. The first four are based on simple criteria and require input of the onset and fully turbulent values which are entered in the *Onset* and *Fully Turbulent* input boxes located below the transition criteria menu. The remaining two are more complex criteria developed for the NASP program, and they do not require input of the onset and fully turbulent values.



**Figure 4.6-7 Transition Criteria Options**

*Running Length*- The first criteria is based on the running length along the streamline. The user must specify transition onset distance and fully turbulent distance in the geometry units.

*Re* - The second criteria is the local Reynolds number calculated based on edge properties. The value at onset and at the fully turbulent condition are input by the user.

*Re, theta* - The third criteria is the local Reynolds number based on the edge properties and the compressible momentum thickness calculated using the Eckert enthalpy method. The value at onset and at the fully turbulent condition are input by the user.

*Re, theta/Me* - The fourth option, the momentum thickness Reynolds number divided by the local edge Mach number, is the most common criteria. The value at onset and at the fully turbulent condition are input by the user.

*BLT-1A* - The NASP BLT-1A criteria is used to predict transition for slender axisymmetric geometries.

*BLT-2D* - The NASP BLT-2D criteria is a planar flow transition criteria developed specifically for aerodynamic analysis methods such as S/HABP.

**4.6.1.6 Boundary Layer Model** - The component may be modeled as either a flat plate or a leading edge by selecting the appropriate option in the boundary layer model option menu shown in Figure 4.6-8. Additional inputs required by the flat plate and leading edge option are discussed respectively in the next two sections.

Boundary Layer Model

Leading edge	<input checked="" type="checkbox"/>
Flat plate	<input type="checkbox"/>

**Figure 4.6-8 Component Modeled as Leading Edge or Flat Plate**

**4.6.1.7 Flat Plate** - As shown in Figure 4.6-9 selecting the flat plate boundary layer model activates the input menus associated with flat plate analysis. These menus include Plate Laminar Method, Plate Turbulent Method and Cone Correction.

Boundary Layer Model

Flat plate	<input checked="" type="checkbox"/>
------------	-------------------------------------

Plate Laminar Method

Eckerts	<input checked="" type="checkbox"/>
---------	-------------------------------------

Plate Turbulent Method

Schultz-Grunow	<input checked="" type="checkbox"/>
----------------	-------------------------------------

Leading Edge Type

Swept cylinder	<input checked="" type="checkbox"/>
----------------	-------------------------------------

Nose Radius (in):

--

Fin sweep angle (deg):

--

Fin elevation angle (deg):

--

L.E. Method

Not swept cylinder	<input checked="" type="checkbox"/>
--------------------	-------------------------------------

Cone Correction

Flat plate	<input checked="" type="checkbox"/>
------------	-------------------------------------

**Figure 4.6-9 Inputs Associated with Flat Plate Methods**

*Plate Laminar Method* - The two available laminar skin friction and aeroheating correlations are the Eckert and the Rho-Mu methods. Both methods are based on the classic Blasius flat plate boundary layer solution corrected with the reference enthalpy compressibility factors. The Eckert method is well known and has been shown to be very accurate.

*Plate Turbulent Method* - The four available turbulent methods are shown in Figure 4.6-10.

Plate Turbulent Method

Schultz-Grunow	<input checked="" type="checkbox"/>
Rho-Mu	<input type="checkbox"/>
Spalding-Chi	<input type="checkbox"/>
White	<input type="checkbox"/>

**Figure 4.6-10 Turbulent Flat Plate Boundary Layer and Aeroheating Methods**

The Schultz-Grunow and the Rho-Mu methods are incompressible formulas that use the reference enthalpy compressibility transformation methods. Spalding-Chi is a well known boundary layer method that is also used in the level 1 viscous analysis as well as other codes such as AEROHEAT and MINIVER. The White method is similar to Spalding-Chi except that it uses a curve fit approximation of the skin friction formula.

*Cone Correction* - As shown in Figure 4.6-11, the component may be analyzed either as a flat plate or a cone. The cone correction flag is used to set the Mangler factor which transforms the solution of two dimensional boundary layer to the axially symmetrical case.

Cone Correction    Flat plate ☒

                              Cone ☐

**Figure 4.6-11 Cone Correction for Flat Plate Analysis**

The flat plate option is selected for components with 2-D flow which is characterized by parallel streamlines. Areas which usually exhibit 2-D flow include ramps, wings, fins, tails and cylinders at zero alpha. The cone option is selected for components with axisymmetric flow which is characterized by divergent streamlines. Axisymmetric areas includes circular noses, ogives, conical fuselages or cylinders at angle of attack.

**4.6.1.8 Leading Edge** - Selecting the leading edge boundary layer model activates the inputs associated with the leading edge method, shown in Figure 4.6-12. If the component is modeled as a leading edge, the user must select one of the leading edge types and input the nose radius. Some of the remaining parameters must be specified depending on which leading edge type was chosen.

Boundary Layer Model    Leading edge ☒

Plate Laminar Method    Eckerts ☒

Plate Turbulent Method    Schultz-Grunow ☒

Leading Edge Type    Swept cylinder ☒

Nose Radius [in]:    1.000

Fin sweep angle (deg):    30.00

Fin elevation angle (deg):    90.00

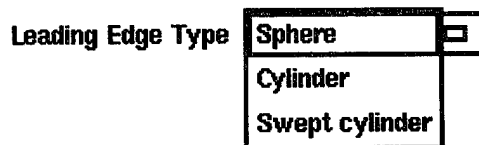
L.E. Method    Not swept cylinder ☐

Cone Correction    Flat plate ☒

**Figure 4.6-12 Inputs Associated with Leading Edge Method**



**Leading Edge Type** - As shown in Figure 4.6-13, the leading edge may be modeled as either a sphere, cylinder or a swept cylinder.



**Figure 4.6-13 Leading Edge Types Available In Level 2 Viscous analysis**

Spherical leading edge is usually modeled at the front of an ogive or a conical forebody. Additional input required is *Nose Radius*.

Cylinders are used to model the leading edge of 2-D forebodies, inlets, or unswept wings, fins and tails. Additional inputs required are *Leading Edge radius* and *Fins Elevation Angle*.

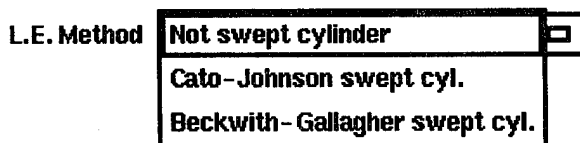
Swept cylinders are commonly found as the leading edge of swept wings, fins, and tails. Additional inputs for the swept cylinder are *Leading Edge Radius*, *Fin Sweep Angle* and the *Fin Elevation Angle*.

**Leading Edge Radius** - The leading edge radius input is required for all the leading edge types. The value is input in the geometry units.

**Fin Sweep Angle** - This value is required for the swept cylinder leading edge type. The sweep angle is zero perpendicular to the vehicle's longitudinal axis and ranges from -90 to 0 degrees for forward sweep and 0 to +90 degrees for aft sweep.

**Fin Elevation Angle** - This value is required for both swept and unswept cylinder leading edge types. The fin elevation angle is measured at its root chord. The angle ranges from 0 degree in the -z direction (e.g., a vertical tail on the bottom centerline) to 180 degree in the +z direction (e.g., a vertical tail on the top centerline).

**Leading Edge Method** - The menu, shown in Figure 4.6-14, is used for selecting the swept cylinder analysis method.



**Figure 4.6-14 Swept Cylinder Analysis Methods**

If the user input zero degree sweep angle for a swept cylinder leading edge type, any of the three methods may be selected. If the sweep angle is nonzero, one of the two swept cylinder methods must be used. The *Not Swept Cylinder* option assumes that the leading edge is an unswept cylinder. With this option, the boundary layer analysis is performed with the same methods used for spherical and unswept cylinder leading edges. These methods include the Lee's method (Reference 8) for laminar flow and the Detra-Hidalgo method (Reference 9) for turbulent flow.

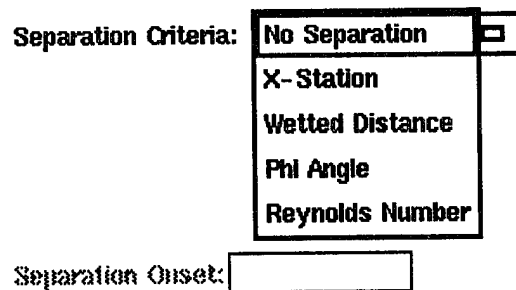
Swept cylinder analysis predict the laminar flow characteristics only. The analysis uses Lee's method with the addition of the sweep angle effect, which is calculated with either the *Cato-Johnson* (Reference 10) or the *Beckwith-Gallagher* swept cylinder method (Reference 11).

**4.6.1.9 Inviscid Analysis for Input to Viscous Routines** - Surface pressures for the level 2 analysis are calculated using the inputs specified from the level 2 viscous analysis window. Inputs for the standard inviscid pressure analysis, shown in Figure 4.6-15, are the same as those used for the inviscid analysis. The details of the inviscid pressure methods and the required input are found in the *Inviscid Analysis* section (4.4).

Windward Method:	<input type="text" value="Tangent-cone"/>	<input type="checkbox"/>
Leeward Method:	<input type="text" value="Newtonian"/>	<input type="checkbox"/>
Dynamic Pressure Ratio:	<input type="text" value="1.00"/>	
Not used:	<input type="text"/>	
Not used:	<input type="text"/>	
Surface Slope Modification Factor:	<input type="text" value="1.0000"/>	
Impact Shock Method:	<input type="text" value="Tangent-cone"/>	<input type="checkbox"/>

**Figure 4.6-15 Inviscid Analysis Input for Level 2 Viscous Analysis**

**4.6.1.10 Separation Criteria** - Level 2 viscous analysis has the capability to model separated flow. Flow separation is based on one of the criteria shown in Figure 4.6-16. After selecting the desired criteria, the user must specify the separation onset value. At each point along the streamline, the local condition based on the selected criteria is evaluated. When the local value equals or exceeds the onset value, the flow at the current point and aft are considered separated. In the separated region, the pressure is set to the value calculated at the point of separation and the skin friction is set to zero.



**Figure 4.6-16 Flow Separation Criteria Menu**

The x-station and the wetted distance onset values are input in the geometry units. Phi angle, the z component of the unit normal, is input in degrees. The Reynolds number is based on the local edge properties and wetted distance. Note that the separation onset input box is not active when no separation is selected.

**4.6.1.11 Print Flag** - Output for the level 2 viscous analysis is controlled by three print flags shown in Figure 4.6-17.

- ☒ **Print Streamline Output**
- ☐ **Print Interpolated Elemental Output**
- ☐ **Print Viscous Geometry/ERF**

**Figure 4.6-17 Level 2 Viscous Analysis Output Formats**

The first flag prints output of predictions along the streamlines. The second flag prints output of predictions after interpolation onto the element centroids. The third flag provides output for use with the VUAERO program. Any combination of the print options may be selected for any component. However, the streamline output flag is global. If the streamline output print flag is specified for at least one component, the streamline result is output for all components.

## 4.6.2 Output

In addition to calculating the viscous contribution to the total forces and moments, the level 2 viscous analysis output includes the following:

- *Skin Friction Coefficient* - For consistency with the level 1 format, the output value is local skin friction coefficient multiplied by the dynamic pressure ratio.
- *Heat Transfer Coefficient* - The value is based on enthalpy rather than temperature to eliminate the variation of the specific heat due to temperature. The heat transfer coefficient has the unit of  $\text{lbm/ft}^2\text{-s}$ .
- *Momentum Thickness* - Output in feet.

- *Displacement Thickness* - Output in feet.
- *Wall Temperature* - The output is either the user specified wall temperature or the calculated radiation equilibrium temperature. Output in degrees Fahrenheit.
- *Wall Pressure* - Output in lbf/ft<sup>2</sup>

**4.6.2.1 Text Output** - Level 2 streamline results and interpolated element results are tabulated in the output file.

*Streamline Results* - An example of the streamline result output is shown in Figure 4.6-18. The streamline results are grouped by the alpha-beta pairs that were analyzed. The first line of each alpha-beta set lists the alpha and beta values and the number of streamlines contained in this group. In the example, there are 36 streamlines in the Alpha = 0 and Beta = 0 set, but only the first streamline is shown. For each streamline point, the wetted distance, local skin friction coefficient, heat transfer coefficient, adiabatic wall enthalpy, wall temperature, wall pressure, and the flow regime are tabulated.

```
Alpha = 0.00 Beta = 0.00 36 Streamlines
Streamline: 1
Wetted Distance (in) Skin Friction Coeff. (---) Heat Transfer Coeff. (lbm/ft2/s) Adiabatic Wall Enthalpy (Btu/lb) Wall Temp. (F) Heat Flux (Btu/ft2-s) Wall Pressure (lbf/ft2) Flow Regime (---)
0.353 0.009489 0.35635E-01 522.8 1253.7 3.29 127.0 laminar
3.535 0.003001 0.11269E-01 522.8 1010.4 1.78 127.0 laminar
15.718 0.000843 0.27999E-02 522.2 692.7 0.67 32.9 laminar
36.379 0.000618 0.20468E-02 523.0 625.1 0.53 32.7 transition
60.404 0.001151 0.37573E-02 531.3 767.5 0.86 32.3 transition
84.428 0.001652 0.53785E-02 538.9 858.7 1.15 32.2 transition
108.455 0.001841 0.59890E-02 541.9 887.4 1.26 33.0 turbulent
132.485 0.001788 0.58178E-02 541.9 880.5 1.23 33.1 turbulent
144.500 0.001765 0.57416E-02 541.9 877.4 1.22 33.1 turbulent
```

**Figure 4.6-18 Streamline Result Output Format**

*Interpolated Element Results*- An example of the output format is shown in Figure 4.5-19. For each component, the level 2 results are placed in the output file right before the skin friction force summation output. The element is identified by the panel and element ID's. For each element, the momentum thickness, displacement thickness, skin friction coefficient, heat transfer coefficient, heat flux and wall temperature have been interpolated from the surrounding streamline points to the centroid of the element. The interpolation procedure provides two error messages. One message, shown in the example for the sixth element of the ninth panel, indicates that the number of streamline points found within the radius of influence exceeds the maximum allowable memory. This is not a fatal error, but the interpolated result may not represent the data closest to element of interest. However, if no streamline point is found within the radius of influence, the interpolation cannot be performed. This is a fatal error, and the program halts.

INTERPOLATED BOUNDARY LAYER PROPERTIES

ALPHA = 0.00

BETA = 0.00

PANEL ID	ELEMENT ID	MOMENTUM THICKNESS (FT)	DISPLACEMENT THICKNESS (FT)	SKIN FRICTION COEFF.	HEAT TRANSFER COEFF. (BTU/FT <sup>2</sup> -S-F)	HEAT FLUX (BTU/FT <sup>2</sup> -S)	WALL TEMP (F)
1	1	0.000484	0.000832	0.005705	0.051878	4.7700	1402.5787
1	2	0.000610	0.001050	0.005479	0.051864	4.7691	1402.5070
4	78	0.000811	0.001392	0.000986	0.003527	0.7934	741.1825
4	79	0.000804	0.001383	0.001008	0.003544	0.7965	742.4889
4	80	0.000802	0.001380	0.001022	0.003564	0.8000	743.8616
8	11	0.007610	0.009899	0.002852	0.007966	1.5106	950.4246
8	12	0.007928	0.010285	0.002869	0.008537	1.5871	967.5329
8	13	0.008212	0.010627	0.002851	0.009126	1.6657	985.1150
9	5	0.013089	0.016885	0.002652	0.008727	1.6283	977.8958
EXCEEDED MAX NUMBER OF DATA POINTS FOR INTERPOLATION							
ONLY MAX POINTS WILL BE USED FOR INTERPOLATION							
9	6	0.013096	0.016893	0.002662	0.008785	1.6356	979.5081
9	7	0.013074	0.016865	0.002673	0.008901	1.6501	982.6766
15	53	0.018576	0.023963	0.001112	0.003352	0.8165	747.2101
15	54	0.018581	0.023970	0.001112	0.003413	0.8273	750.91771

Figure 4.6-19 Interpolated Element Results Format

**4.6.2.2 Graphical Output** - The graphical output consists of a type 3 geometry file (VISCOUS.TP3) and an element result file (VISCOUS.ERF) which are used with VUAERO for graphical data analysis. The element result file contains seven boundary layer properties:

1. Momentum Thickness (ft)
2. Displacement Thickness (ft)
3. Pressure Coefficient (dimensionless)
4. Skin Friction Coefficient (dimensionless)
5. Heat Transfer Coefficient (lbm/ft<sup>2</sup>-s)
6. Heat Flux (BTU/ft<sup>2</sup>-s)
7. Wall Temperature (°F)

A detailed tutorial for using VUAERO including the format for the type 3 geometry and the element results can be found in the *VUAERO User's Manual*. In VUAERO, the element result to be displayed is identified as a column number by the user. In the element result file, the value of the properties for each element are arranged in the following format:

VAL1 (I=1, NAB) , VAL2 (I=1, NAB) , . . . VAL7 (I=1, NAB)

where VAL1 through VAL7 represents the momentum thickness through wall temperature respectively, and NAB is the number of cases calculated. Therefore to display the desired property of a particular case, the user must calculate the appropriate column number. For the VISCOUS.ERF file, the column number is determined with the following formula:

$$(N-1)NAB + IAB$$

where N is the ID number (1-7) of the desired property, and IAB is the desired case. As an example, the column number of the heat transfer coefficient (N = 5) for the second (IAB = 2) of the three cases analyzed (NAB = 3) is calculated using the above formula:

$$(5 - 1) * 3 + 2 = 14$$

## 5. TRIM ANALYSIS

The trim program calculates the trimmed aerodynamics in a multistep process. First, the program calculates the moments at the user specified center of gravity (cg) location. Once the moments are known for the cg location, the deflection required to produce zero pitching moment is determined by interpolation using a polynomial fit. The same polynomial interpolation routine is then used to determine the trim coefficients for the given trim deflection. If the extrapolation flag is turned off, the program checks if the calculated trim deflection angle is within the bounds of the available data. If the trim deflection angle exceeds the data limits, a message is written to the output file.

### 5.1 Trim Inputs

The first step to calculate trimmed aerodynamic coefficients is to set up a S/HABP analysis. The S/HABP data required to run the trim program are the forces and moments on the vehicle as a function of angle of attack for multiple deflections. The trim program is set up to handle multiple cases which can consist of changes in Mach number, configuration, etc. Once the desired cases have been defined, the summations must be defined such that each summation includes the total forces and moments for the configuration at a given deflection. A minimum of two summations/deflections are required to calculate the trimmed aerodynamics. The summations may be defined in any order such that the first summation does not need to correspond to the most negative deflection. Only summations (i.e., deflection angles) defined for a given case can be used to calculate trimmed coefficients for that case.

The trim program can be executed as a stand-alone program or can be accessed from the VECC Graphical User Interface (GUI). When the trim program is accessed from the GUI, the inputs are entered using the "Trim Option" window as shown in Figure 5.1-1. When executed as a stand-alone program, the trim program interactively prompts the user for the required inputs.

The inputs needed to run the trim program include the desired S/HABP cases, the summations and corresponding deflections, the center of gravity (cg) position about which the trim coefficients are calculated, and a flag to allow extrapolation.

Available Cases		Select Summations/ Enter deflection	
Case 1		Summation 1	-10.00
Case 2		Summation 2	0.00
Case 3		Summation 3	5.00
		Summation 4	10.00

X, Y, Z Cg Locations to Trim			
-100.000	0.000	0.000	
-120.000	0.000	0.000	
-140.000	0.000	0.000	

Allow Extrapolation: ☐ No ☒ Yes

OK Apply Cancel Help

Figure 5.1-1 Trim Option Window

### 5.1.1 Case Selection

A list of available cases is shown in the upper left part of the trim window. The user can select multiple cases to be trimmed within a single execution of the trim program. There is no restriction on the differences between cases. Cases are independent of each other except for the extrapolation flag and the cg positions which are common to all cases.

### 5.1.2 Summation/Deflection Input

A list of summations for the selected case is shown in the upper right corner of the trim window. The user selects the desired summations and then enters the corresponding deflections. The deflection angle in degrees is entered on the same line and immediately after the selected summation. The summations/deflections can be input in any order to simplify the process of setting up the necessary S/HABP runs. The deflection angle is input for the left control surface with a positive deflection being determined by the right hand rule. Place your right thumb pointing from tip to root and your fingers will wrap in the direction of a positive deflection. For example consider a left horizontal surface, pointing your right thumb from tip to root, your fingers wrap in the leading edge up direction (or trailing edge down depending on your point of view). Thus a positive deflection for a horizontal surface is trailing edge down. A positive deflection of a vertical tail is trailing edge left when looking from the rear. A positive aileron deflection places the left trailing edge down and the right trailing edge up thus generating a positive rolling moment.



**Note:** The user must input a deflection angle for each summation and a minimum of two summations are required to calculate the trim aerodynamic data.

### **5.1.3 Extrapolation Flag**

The extrapolation flag allows the user to turn on/off extrapolation. If extrapolation is turned off and the trimmed deflection is outside the bounds of the data a message will be written to the output file informing the user of the fact. The extrapolation flag applies to all cases trimmed.

### **5.1.4 Center of Gravity Input**

Multiple cg positions can be entered to calculate trim aerodynamics, such that the untrimmed data are shifted to the input cg before the trimmed coefficients are calculated. The user enters the x,y and z coordinates for each cg position on a separate line separated by a space or a comma. Additional cg locations can be entered anywhere in the list. Center of gravity locations can be deleted by deleting the values on a single line (Note: The blank line will remain until the OK button is clicked.). Modifications can be made to a cg location by moving the pointer to the line displaying the cg position desired and editing the values.

## **5.2 Trimmed Aerodynamic Output**

The output from the trim program is shown in Figure 5.2-1 and consists of the trim deflection angle and the aerodynamic coefficients for each alpha-beta combination, center of gravity location and each case. The output format is similar to the S/HABP untrimmed force and moment output and has been kept to 80 columns to ease printing of the data. Each case is started with a case header that includes the case number and the case title from S/HABP. Also included in the case header are the flight conditions, reference quantities and the selected summations. Within each case, the trimmed aerodynamic data are printed out as a function of angle of attack for each center of gravity location.

\*\* S/HABP TRIM AERODYNAMICS OUTPUT \*\*

LONGITUDINAL TRIM CASE # 1 Case 1

MACH NUMBER	VELOCITY [FT/SEC]	REYNOLDS # [1/FT]	ALTITUDE [FT]
3.00	3349.3	2.1296E+07	0.0

SREF	MAC	SPAN	XCG	YCG	ZCG
1.0000E+02	100.000	10.000	-50.000	0.000	0.000

SELECTED SUMMATIONS	DEFLECTION ANGLE(DEG)
1	-20.00
2	-10.00
3	0.00
4	10.00
5	20.00

TRIM DATA FOR XCG = -40.000 YCG = 0.000 ZCG = 0.000

ALPHA	BETA	DELTA	C N C Y	C A C LN	C L C LL	C D	L/D
0.0	0.0	0.00	0.00000	0.12847	0.00000	0.12847	0.00000
2.0	0.0	-2.14	0.16642	0.12872	0.16183	0.13445	1.20365
4.0	0.0	-4.46	0.34018	0.13240	0.33011	0.15581	2.11871
6.0	0.0	-6.83	0.51785	0.14179	0.50019	0.19515	2.56316
8.0	0.0	-9.24	0.69815	0.15708	0.66949	0.25272	2.64916
10.0	0.0	-11.63	0.88146	0.17630	0.83746	0.32668	2.56351
12.0	0.0	-13.99	1.08859	0.19570	1.02412	0.41776	2.45147
14.0	0.0	-16.43	1.30396	0.21908	1.21222	0.52802	2.29578
16.0	0.0	-18.82	1.52208	0.24465	1.39568	0.65472	2.13174
18.0	0.0		** VALUE NOT WITHIN BOUNDS (NO EXTRAPOLATION)	**			
20.0	0.0		** VALUE NOT WITHIN BOUNDS (NO EXTRAPOLATION)	**			

Figure 5.2-1 Sample Trim Output File

## 6. RESULTS DISPLAY

There are two options for graphical display of results in the VECC system. The first option is to use the line plotting program *hplot*. This line plotting module allows the user to create and print report quality plots. Data from S/HABP Mark V as well as the trim module output may be plotted. The second option for graphical results display is to view the streamline traces generated by the QUADSTREAM module. Both of these methods are described in the following sections.

### 6.1 Line Plotting

The line plotting module, *hplot*, is an interactive color graphing program which displays results from the vehicle analysis. Hplot produces various styles of X-Y graphs based on data contained in the S/HABP output files (either the ".out" file for untrimmed aerodynamics or the ".trm" file for trimmed data). The line plotting module is started by selecting either the **Untrimmed Aero...** or the **Trimmed Aero...** item from the **Results** menu in the VECC main menu bar shown in Figure 6.1-1.

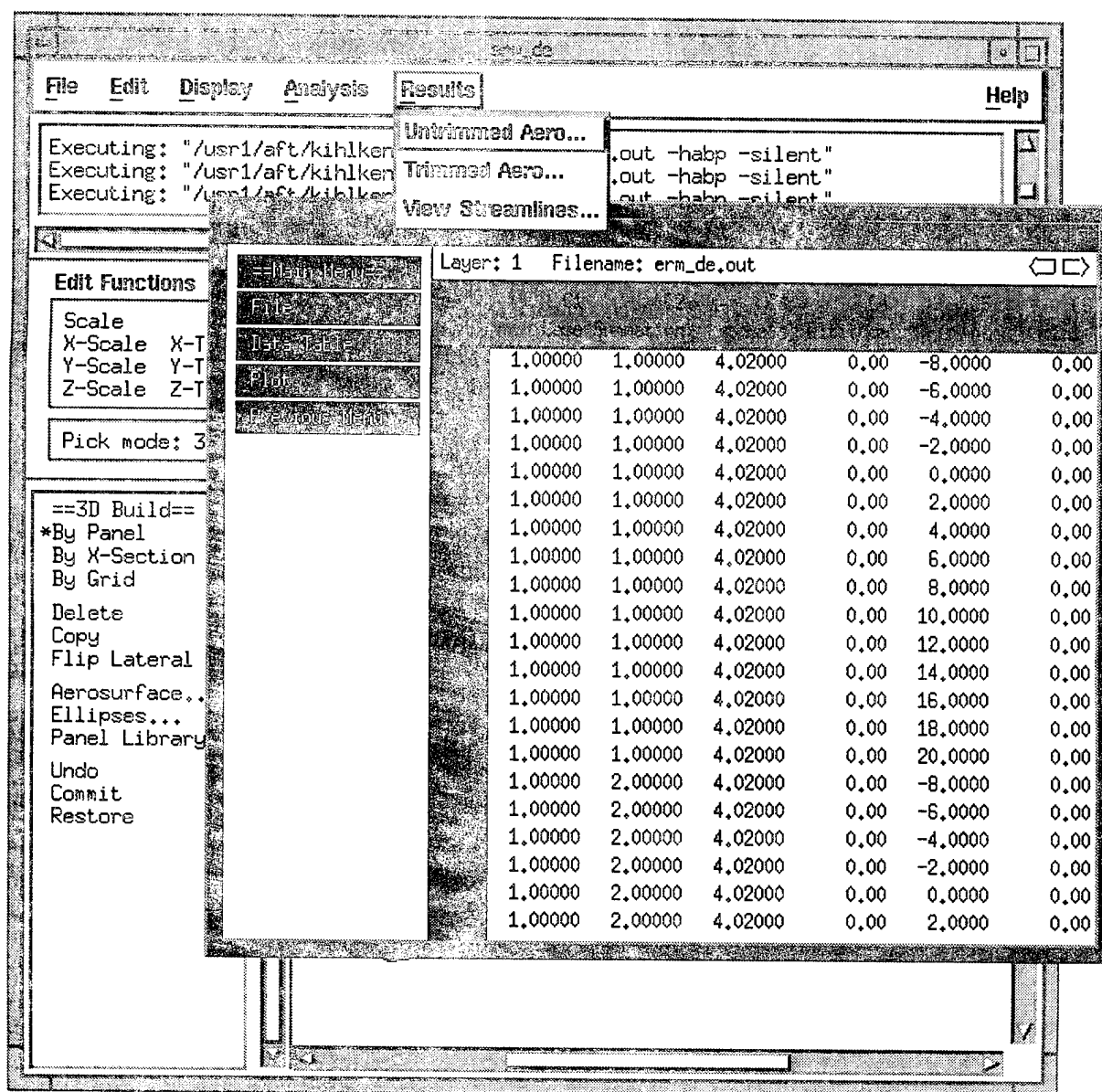


Figure 6.1-1 Hplot Line Plotting Program Invoked from VECC

Hplot is opened with the aerodynamic results from the current output file. Plots are defined by selecting pre-defined quickplots or by selecting rows and columns from the data table. Additional formatting information such as titles, axis scaling, and line styles are specified using a system of menus which are selected with the mouse. This plot format information can be saved in a hplot input file using the File>Save menu. When a plot format is saved in this way, it is referred to as a quickplot and can be recalled from the Plot>Quickplot menu. Several quickplots have been customized from the S/HABP output.

### 6.1.1 Hplot User Interface

Hplot uses a specialized graphical user interface rather than the Motif interface found in the VECC modeler. Program inputs are made using menus which appear along the left edge of each window, by responding to the dialog box using the keyboard, and by dragging elements with the mouse. The left mouse button is used to select elements on the screen. The middle mouse button is used to drag and resize items or to add rows and columns to a data table. The right mouse button is not used by hplot.

Multiple windows are displayed which can be independently moved, resized, or iconified using the workstations window manager. One window displays the data in a spreadsheet format. Other windows display X-Y plots of selected data. The mouse can be used in any window to control the display using the hplot menu system.

**6.1.1.1 Menus** - Menus are displayed on the left side of each window and can be activated using the left mouse button. Some menu selections result in additional submenus which are displayed below the main menu. Hplot always displays the main menu and the last submenu in each window. Selecting Previous Menu will remove the last submenu and revert to the next higher level menu. After selecting Previous Menu several times, only the main menu will remain.

Some menu items represent On/Off options which are toggled when pressed. A toggle which is currently turned on has an asterisk in front of the item.

Certain menu selections may not be available at all times. For example, on a black and white monitor, color options will be inactive. Inactive menu selections will be shown in grey (on monochrome monitors they are lined out).

When a menu contains more selections than can be fit into the window, a box labeled "...more..." is displayed at the bottom of the menu. Selecting this box rolls the menu upward so that the remaining selections become visible. As an alternative, the window can be enlarged to allow more of the menu choices to fit.

**6.1.1.2 Dialog Boxes** - A dialog box is a rectangular area with a text prompt and a region to enter text with the keyboard. Dialogs appear in the center of a window whenever an option is selected which requires additional text input. When a dialog box appears, enter the value and press enter to complete the command. Pressing escape or clicking with the mouse cancels the dialog box.

### 6.1.2 Data Window

The primary hplot window contains the data table along with the filename and the main menu as shown in Figure 6.1-1. The data window is used to view or edit the input data and to select rows and columns for plotting. The data table can have multiple layers, each layer containing many rows and columns.

For large input files, the entire data table will not be visible in the data window. Enlarging the window also allows viewing more of the table. To shift other rows or columns into view, use the right, left, up, down, home, end, previous screen and next screen keys.

The numeric display format for the data values can be changed by clicking on the column number at the top of a column with the left mouse button. A menu of formatting options appears below the main menu. You may enter the column width, number of decimals, or select scientific notation.

The Data Window also contains a small icon to control the use of color. Clicking on the small box above the row numbers will toggle the hplot windows between color and monochrome mode. You may need to turn color off when making hardcopies on some printers.

**6.1.2.1 Data Layers** - The hplot program can process multiple files of results data at the same time. When multiple files are used, each file is placed on a separate layer in the data table. Only one layer can be displayed at a time however. To switch to another layer, press the left and right arrows in the top right corner of the window.

Each data layer is independent from all other layers and can contain different items in each row and column. The filename for a layer can be changed by clicking on it with the left mouse button and entering the new filename at the prompt.

**6.1.2.2 Editing a Data Table** - To change the value stored in a data cell, click on the desired cell with the left mouse button. A dialog box will appear showing the current value. Enter the new value and press <return>. When a data cell is changed, any graphs using that column are immediately redrawn with the new values.

The column titles can also be changed by selecting a title with the left mouse button and entering a new title in the dialog box.

To expand the number of rows and columns in the data table, click the middle mouse button after the last row or last column. The new row or column will be filled with zeros. You can then enter values as described above.

**6.1.2.3 Data Window Menus** - The main hplot menu always appears in the top left corner of the data window. Use these menu options to manipulate the format and contents of the Data Window and to create plots. The following sections describe each option on the Main Menu.

**File Menu** - The File Menu is used to quit the hplot program and to control file input and output.

**New Layer** Opens a new data layer that is empty and ready for data input. Rows and columns can be created in this layer by clicking in the table with the middle mouse button and editing the data as described in Section 6.1.2.2, or an existing data file can be read into the layer using the include option.

**Open File** Opens a new layer and reads in an existing data file or quickplot file. The user will be prompted for the filename to be read. Data files can either be simple column data files (containing test data or predictions from another program) or other S/HABP output files. The filename is entered with the following format:

**filename [arguments]**

Any file extensions must be specified as part of the filename. The filename specified should contain either the data to be plotted or the hplot quickplot commands for creating plots. The optional command line arguments can be one or more of the following:

- |                 |  |
|-----------------|--|
| -habp           | This argument must be used when reading in another S/HABP untrimmed aerodynamic output file (i.e., the ".out" file) for comparing predictions.   |
| -trim           | This argument must be used when reading in results from an S/HABP trim analysis (i.e., the ".trm" file).   |
| -batch          | Runs the program in batch mode. In this mode, an hplot input file is read, and a postscript output file is created containing each of the quickplots. The postscript output can then be sent to a printer for hardcopies. The user can use the batch mode from any text terminal; one does not need a X-Window terminal. |
| -display node:0 | The plot windows are displayed on the specified node. The default node is specified by the environment variable "DISPLAY." This argument is only needed to redirect the windows to another workstation.  |

- help                      Prints a summary of each command line argument.
- notable                Causes the data table to not be displayed. This option should only be used if the *openwindow* keyword is specified in a quickplot file. When the data table is not displayed, you are not able to specify additional plots.

**Close Layer** Deletes a data table and closes the layer.

**Erase Layer** Deletes all data from a table and leaves the layer open for new data input.

**Include File** Appends another data file at the end of the data table in the current layer. The user will be prompted for a filename to be read. A conversion specification similar to the command line arguments must be specified along with the filename (either -trim or -habp as discussed above) if the file is not in the free field format or is not a quickplot file.

**Save Plots** Gives the user the opportunity to save new plots currently on the screen in a quickplot file. All open plots and the predefined quickplots are written to one quickplot file that can be used in future hplot sessions to recreate the plots with the same format (see example below). However, a quickplot file does not contain any data but obtains them from a data file. Therefore, in order to recreate the exact same plots, the data file cannot be altered.

**Quit** All windows are closed and hplot terminates.

**Example of Saving Quickplots** - The procedure for saving and retrieving user created plots are illustrated with the following steps:

1. Create the desired plots following the procedures described in this section.
2. Select **Save Plot** menu item discussed above to save all open plots to a quickplot file. At which time, the user will be asked to name the quickplot file. This file contains hplot instructions such as title formats, data filename, and location of data within the file.
3. The quickplot file does not contain the actual data. The data is contained within the S/HABP output file. To recreate the exact same plot, the data file must remain unchanged. Note: The quickplot file is a text file and not the actual picture of the plot itself. To save a plot for later printing, choose the **Write PS...** item in the **Tools** menu of the plot window (see Section 6.1.3.1).
4. Your saved plots can be retrieved by selecting **Open...** from the **File** menu in the hplot window. The user will be prompted for the name of the quickplot file created in Step 2.



5. The user's quickplots will be listed after the predefined quickplots and accessible as described in Section 6.1.2.3. If you ran more than one case in S/HABP, your new quickplots will be listed along with the predefined quickplots of the last case (you are prompted for the case number before you select the desired quickplot).

**Data Table Menu** - The Data Table Menu is used to modify certain rows and columns of the data table in the current layer of the Data Window.

**Math** The math function allows evaluating complex functions on the input data. After selecting *Math*, a FORTRAN-like expression (which can include column values) can be entered in the dialog box. A new column is then formed from the result of the expression. The expression can use combinations of any of the following operators and functions: +, -, \*, /, \*\*, sin, cos, tan, sind, cosd, tand, asin, acos, atan, int, abs, log10, log, exp, and sqrt. The data columns are designated by their number, i.e. C1, C2, etc. The row numbers also can be used by specifying column zero (C0). The following are some simple examples.

- 1)  $C3 + C5$
- 2)  $C2 = C5 * \sin(C7)$
- 3)  $C2 = C2 / 1000$

In Example 1, the result is placed in a new column at the end of the table. In Examples 2 and 3, the result is placed in column 2. When data columns are changed, any graphs using that column are immediately redrawn with the new values.

**Delete Rows** Asks for a starting row, ending row, and a skip count for rows to be deleted. Separate these values with either a space, dash, or comma. The skip count will skip that number of rows after the first row before selecting the next row. For example, a "2" will skip every other row. Note that this option deletes all columns for the specified rows and cannot be undone. Also note that all three arguments need not be specified. For example, to delete only one row, it is sufficient to only specify that row number.

**Delete Column** A submenu with the current column headings is displayed. The selected column is deleted.

**Plot Menu** - The Plot Menu is used to select data sets to be graphed. The user can create a graph using a quickplot or by manually picking columns. Hplot also has the capability of overlaying a new data set on an existing graph.

Selecting *Quickplot* on the Plot Menu will display another menu of all the cases that have been analyzed. After selecting a case, the user may select from several quickplots that have been

customized for S/HABP. Which quickplots are available depends on the derivative options selected when running S/HABP. All of the predefined quickplots are listed below:

- CL vs AOA
- CD vs AOA
- L/D vs AOA
- CM vs AOA
- CN vs AOA
- CA vs AOA
- CMA vs AOA
- CND vs AOA
- CMD vs AOA
- CMQ vs AOA
- CLNB vs AOA
- CLLB vs AOA
- CLND vs AOA
- CLLD vs AOA
- CLNR vs AOA
- CLLP vs AOA

To overlay data on an existing plot, select the *Overlay Data* option on the Plot Menu. Then select the existing plot on which to overlay the new data set and define the data set in the same manner described below for a new plot.

To create a new plot from the data table, select *New Plot*. This will allow for the selection of data sets to be plotted. Each data set consists of a column for the horizontal axis, columns for the vertical axes, optional constraints, a skip count (the default is 1), and left/right vertical axis selection. When selecting columns, the user first selects the horizontal axis and then the vertical axis. An asterisk will appear to the left of each vertical column selected. If the column is clicked a second time, the asterisk disappears and the column is deselected. The row numbers also can be used for either the vertical or horizontal axis. Constraints are very important when generating a new plot. An example is given below (after description of the five *New Plot* menu items) of how constraints may be used.

The menu for selecting a data set is shown below. The first five options will be inactive until a horizontal column is selected.

**Done** Concludes a data set selection. A new graph will be created in a Plot Window.

**Overlay** Allows data sets with different horizontal and vertical columns, constraints, skip values, and left/right axis selections to be graphed on the same plot. For overlays with similar horizontal columns, constraints, etc., it is not necessary to use this option since picking multiple vertical columns on one selection will create a new overlay for each vertical column selected. Each graph can have as many as 12 overlays.

**Constraints** Limits the number of rows which are plotted. Each constraint selects rows in which the selected column is Less Than, Equal To, or Greater Than the constraint value. Multiple constraints can be used on the same data set, however the constraints must refer to columns on layer one. If constraints are not used, all data in a column is treated as a single curve. This can lead to confusing plots if several summations and/or several cases have been run since the last point of one summation will be connected to the first point of the next summation until all data of that column has been plotted. To avoid this, use the constraint *For Each ...* as discussed in the example below. If using the *For Each...* constraint, be sure to specify the other constraints before specifying the *For Each...* constraint because no other constraints may be selected after selecting *For Each....*

**Skip Rows** Normally every row in a data set is graphed. Entering a skip count will skip rows, limiting the data values shown on the graph. For example, a skip count of 5 will plot every fifth row.

**Use Right Axis** Positions the vertical axis to the right of the graph. The default is to place the vertical axis on the left side. Hplot allows using both the left and right axes on the same graph to plot multiple columns with incompatible scales. To do this, select the columns which use the left axis, select *overlay*, then select the columns which use the right axis.

**Example of Using Constraints** - Constraints are usually required to create the desired plot. All of the predefined quickplots for example use constraints to plot each of the summations in a given S/HABP case. This example illustrates how to use constraints to plot the second summation for all cases run in S/HABP.

1. Select *Plot* and then *New Plot* from the main menu.
2. Pick the variable for the horizontal axis by clicking on the variable name.
3. Pick the variable for the vertical axis by clicking on the variable name.
4. Select *Constraints>*.
5. Click on the word *Summation*.
6. Select the *Equal To...* the menu item.
7. A dialog box appears into which you will type "2" (for the second summation) and hit enter.
8. Select *Constraints>* again.
9. Click on the word *Case*.
10. Select the *For Each...* menu item.
11. Select the *Done* menu item and your new plot will appear.

### 6.1.3 Hplot Windows

This section describes the operations which are available in an hplot plot window. A plot window is created each time a new graph is created making it possible to view many plots at the same time. Figure 6.1-2 shows a typical plot.

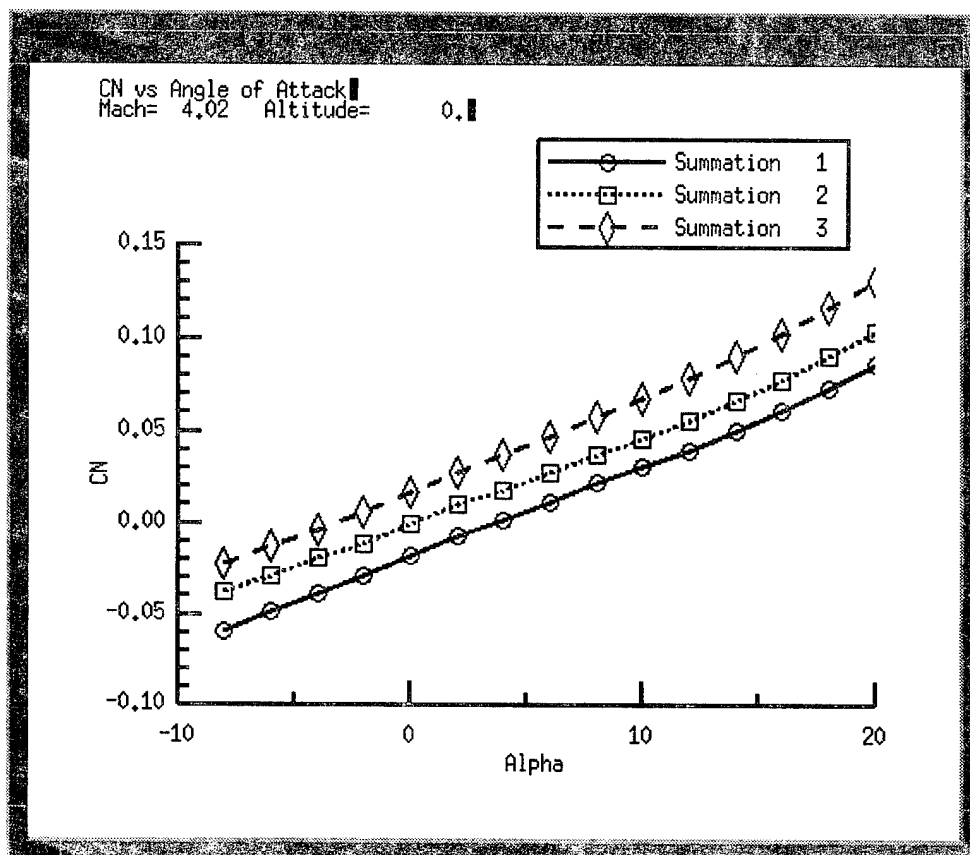


Figure 6.1-2 Sample Plot from a Customized Quickplot

**6.1.3.1 Tools Menu** - The Tools Menu is displayed by clicking in the plot background with the left mouse button. It is used to add entities to a graph and to output a plot to a file or printer.

**New Title** Adds a new title to a plot window. A text cursor will appear on the screen allowing you to select the location for the new title. The new title is then typed into the dialog box.

**New Line** Allows for the creation of new lines on the plot. The cursor becomes a pencil which is used to draw lines in the graph window. Section 6.1.3.2 explains how to add lines to a graph window.

**Paste** Pastes the most recently copied title onto the plot.

**Show Legend** Toggles the plot legend on or off.

**Plot Frame** Toggles the plot frame on or off. The plot frame is a line drawn around the data region.

**Date/Time** Toggles the date/time display on or off.

**Classification** Creates a classification label upon selection of one of the following: UNCLASSIFIED, CONFIDENTIAL, SECRET, TOP SECRET. The label will appear in the top center and bottom center of the plot and can be moved or modified as a title after it is created. The classification label can be treated as normal titles and can be repositioned or otherwise modified as described in Section 6.1.3.2. If you want to change the classification label, simply select it by clicking on it with the left mouse button and deleting it before creating the new classification label.

**Print Plot** Produces a hardcopy of the Plot Window. For more information on printing a graph, see Section 6.1.4.

**Write PS** Writes the plot to an encapsulated postscript file.

**Delete Plot** The plot and Plot Window are deleted from the screen.

**6.1.3.2 Customizing a Graph** - Plots can be customized by rearranging the axes, adding titles and lines, or changing colors and line styles.

The axes can be resized by dragging their endpoints with the **middle** mouse button. An axis can be moved by dragging at its center. In a similar way, titles and the legend can also be repositioned by dragging them with the **middle** mouse button.

**Horizontal and Vertical Axes** - Every graph has a horizontal axis and either 1 or 2 vertical axes. By default, the axes are automatically scaled to produce even increments for the selected data sets. Any of the parameters can be changed for an axis, however, by selecting the axis with the left mouse button. If the values plotted are very small, hplot automatically adjusts the axis label and axis values to reduce the number of digits required for the numeric axis. If for example the derivative  $Cm\delta$  is 0.00001, the axis label will read "CMD, x 0.001", and data point will be plotted at a value of 0.01. Thus, the value read from the plot is multiplied by the factor given in the axis label to get the actual value.

**Numeric Axis Menu** - When a numeric axis is selected on a plot, the Axis Menu appears on the plot

**Start Value** Limits the axis, and therefore the graph, to a starting value.

**End Value** Limits the axis, and therefore the graph, to an ending value.

**Increment** Changes the spacing of the major tick marks on the axis.

**Subticks** Specifies how many subdivisions appear between the main ticks which are labeled with numbers.

**Format** Asks for a character string to format the numbers on the axis. Any valid "C" language conversion string for a floating point number may be used. The format statement is used to change the number of decimal places and spaces allotted for the number. The form of the statement is "*%width.precision*" where *width* is the total width of the field in characters and *precision* is the number of digits to the right of the decimal point. If the width specified in the format statement isn't large enough for the number, the field will be extended to whatever width is necessary. The default format is "%4.0f." The following examples illustrate the floating point format.

<u>Format</u>	<u>Output</u>	<u>Description</u>
%4.0f	100	no decimal places
%4.1f	100.0	one decimal place, width is extended
%2.0f	100	no decimal places, width is extended
%2f	100.000000	default number of decimal places is used and width is extended
%f	100.000000	default width and number of decimal places are used

**Font** Changes the font of the labels on the axis. The font can be changed to Courier, Times (20, 14, 14i, 24i, 32, and 38 point), and Helvetica (14, 20, 20b, 24, and 34 point). The "i" following the size will italicize the title, and the "b" will print the title in bold.

**Major Grid** Draws a bold grid line across the graph at every large tick mark.

**Minor Grid** Draws a grid line across the graph at every tick mark.

**Bold Lines** Makes the axis, tick marks, and major grid lines appear bolder.

**Use Log Scale** Creates a log scale axis (if possible). The log scale option cannot be used if any of the data values are less than or equal to zero.

**Scale Factor** Scales the data by the value specified.

**Customizing Graph Data Lines** - Each column plotted is called a data set and appears in the plot legend. By default, data sets are plotted with solid lines and symbols of various colors. If a

data set contains more than 30 data points, the symbols will not be shown. Each data set can be individually customized by picking the data line in the legend with the left mouse button. The data display menu for that data set then appears .

**Line Style** Allows you to select from either solid or dashed line styles.

**Symbols** Allows selecting a symbol to be drawn at each data point.

**Color** Allows you to select a color for the data set. This option will be lined out on a monochrome monitor.

**Smooth Data** Corrects the line on the plot to curve between the data points instead of plotting straight lines. It cannot use data in which more than one value is used per horizontal axis coordinate. Figure 6.1-3 shows an example of a line which has been smoothed.

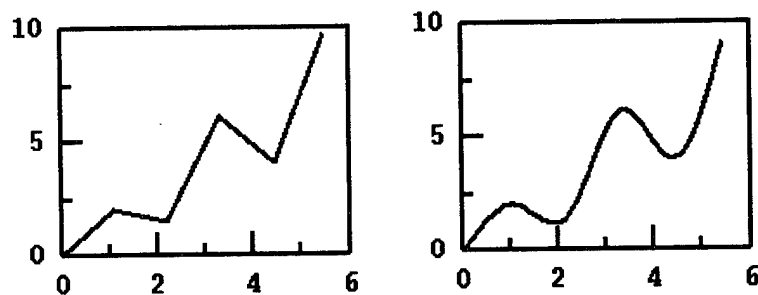


Figure 6.1-3 Effect of Smoothing Data Plots

**Delete Line** Removes the data set from the graph.

**Customizing Titles** - A graph can have many titles with various fonts and colors. By default, each graph is given one title for each axis showing the column name. New titles are added using the Tools Menu. Existing titles can be modified by selecting them with the left mouse button. When this is done, the Title Menu is displayed in the Plot Window.

**Text** Changes the text in the title. Enter the new text in the dialog box.

**Font** Can be changed to Courier, Times (20, 14, 14i, 24i, 32, and 38 point), and Helvetica (14, 20, 20b, 24, and 34 point). The "i" following the size will italicize the title, and the "b" will print the title in bold.

**Color** Changes the color of the title. This option will be lined out on a monochrome monitor.

**Underline** Underlines the title.

**Rotate Vertical** Rotates the title from the first character upward to a vertical position.

**Delete Title** Permanently deletes the selected title from the plot.

**Copy Title** Copies the title into the paste buffer. After the title is copied, it can be pasted onto a plot by using the Paste option on the Plot Tools Menu.

**Cancel Menu** Removes the menu from the screen.

**User-Created Lines** - Each graph can include lines and arrows of various shapes, colors, and styles. New lines are added by selecting *Tools>New Line*. Then place the pencil cursor at the beginning of the line, click the left mouse button, and draw the line to the desired location. Click the mouse button again to end the line. Holding the shift key down allows multiple line segments to be drawn. Holding the alternate key while drawing forces the line segment to be either vertical or horizontal.

Line segments created with the *Tools>New Line* menu can be moved by dragging the line using the middle mouse button. To change the shape of the line, drag either endpoint of the line segment with the middle mouse button. Holding the alternate key while dragging forces the line segment to be either vertical or horizontal.

Arrows may be added to the beginning or end of any line by selecting the line with the left mouse button, then picking *Arrows*.

**Legend** - The legend is a rectangular area showing each data column plotted and its corresponding line style, color, and symbol. The legend may be moved to any location in the window by dragging it with the middle mouse button. To remove the legend from the plot, toggle the *Tools>Show Legend* menu. By clicking on the legend (anywhere other than a data line segment) with the left mouse button, the legend menu will be displayed with the following options.

**Text** Allows the label for the data set to be changed. The default label is the column title.

**Font** Allows the font of the data set in the legend to be changed. The available fonts are Courier, Times (20, 14, 14i, 24i, 32, and 38 point), and Helvetica (14, 20, 20b, 24, and 34 point). The “i” and “b” following the size are for italics and bold, respectively. The default is Times 20.



**Border** A box is drawn around the legend.

**Date/Time Stamp** - The date/time stamp consists of two separate entities that can both be treated as regular titles after they are created. The first line of the date/time stamp is the date, and the second line is the time. When moving the date/time stamp, both entities will be moved together. All title operations can be performed on the separate parts of the date/time stamp except for rotation. If one part of the date/time stamp is deleted, the whole stamp will be deleted.

#### 6.1.4 Print Graphs

Due to the wide variety of workstations and printers, there are many ways to make hardcopies of hplot graphs. Some workstations have screen copy facilities which can copy any region or window on the screen. See the workstation documentation for the appropriate procedure.

Hplot can be configured to automatically print plots by executing a printing program named `ps_print`. The shell script is executed when *Tools>Print Plot* is selected.

The printing program should be modified by the system administrator so that it includes whatever commands are necessary to produce a hardcopy on the system printer. It is executed with one argument which is the name of an encapsulated postscript file. It must be located in the normal search path for executable programs.

Postscript files are also useful for importing into some desktop publishing software. Select Write PS from the Print menu to create a postscript file.

#### 6.1.5 Hplot Plotting Limitations

The following limits are established by internal array sizes when the hplot program is compiled.

Maximum number of data layers .....	20
Maximum number of data columns .....	50
Maximum number of data rows .....	no limit
Maximum number of plot windows .....	10
Maximum number of titles on all plots .....	500
Maximum number of quickplots .....	100
Maximum number of lines created on all plots .....	50
Maximum number of segments per created line .....	5
Maximum number of data sets per plot .....	20
Maximum number of data set constraints .....	6

Additional limitations:

- Constraints only can be associated with Layer 1.

- Only 1 plot can be created in a Plot Window.
- Due to X Window implementation limitations, some workstations do not have all fonts available or may display fonts differently than the menu description.

## 6.2 View Streamlines

Streamlines may be displayed in the VECC window using the menu and dialog box shown in Figure 6.2-1.

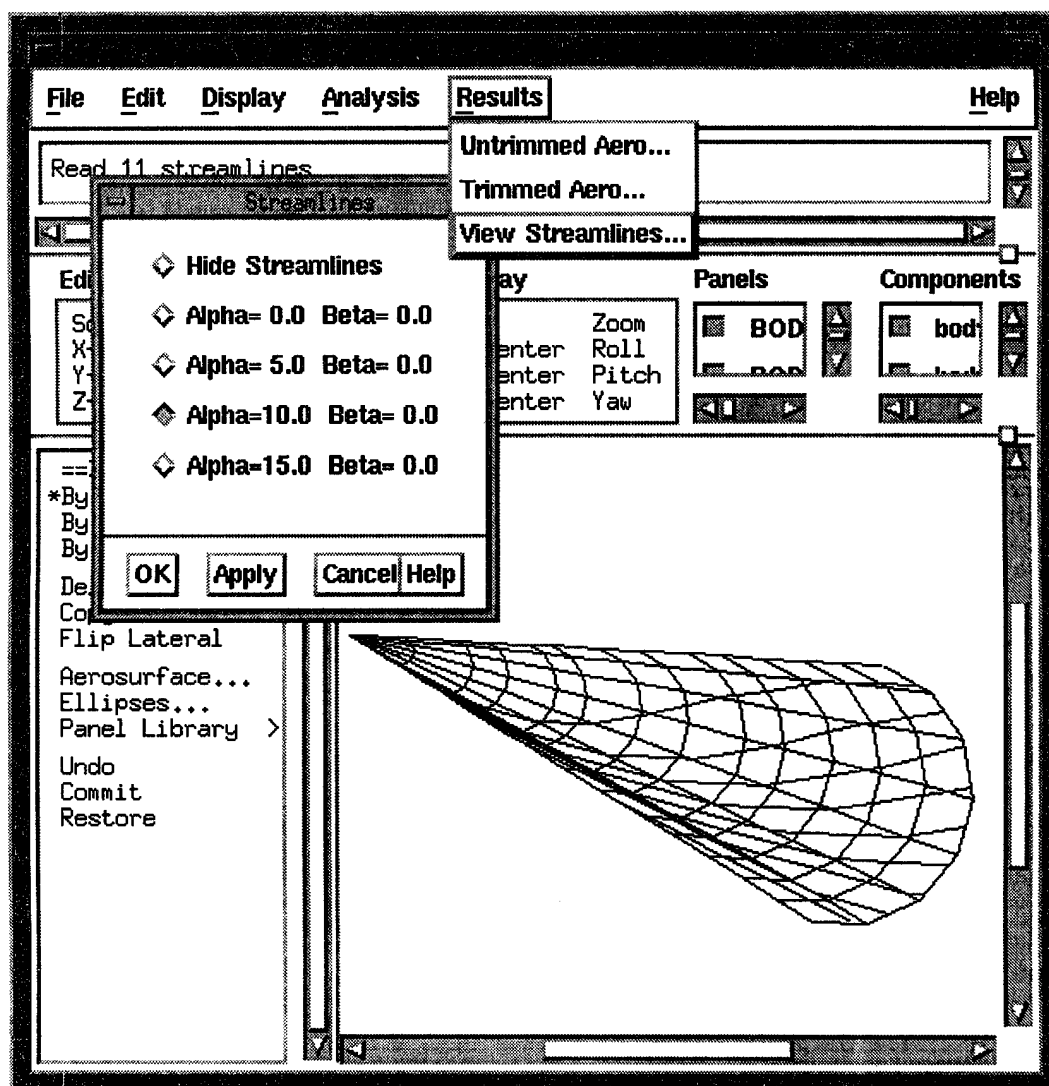


Figure 6.2-1 Display, Menu and Dialog box Used for Streamline Viewing

Results from the streamline analysis may be viewed by selecting the **View Streamlines...** item from the **Results** menu. This action invokes the *Streamlines* dialog box which lists the flight conditions that have been analyzed for streamlines. These conditions, identified by the alpha-beta pairs, were specified in the *Quadstream* window during the streamline input procedure prior to the streamline analysis. Each condition may be viewed one at a time by selecting the desired alpha-beta pair in the dialog box and clicking the **Apply** button. Streamlines may be removed by selecting the **Hide Streamlines...** option.

## 7. REFERENCES

1. Gentry, A. E., Smyth, D. N., and Oliver, W. R., "The Mark IV Supersonic-Hypersonic Arbitrary Body Program," *AFFDL-TR-73-159*, Nov 1973, Volumes I, II, III.
2. Jenn, A. A., Matthews, C. D., Wang, P. K., "Missile Supersonic/Hypersonic Aerodynamics Final Report," *WL-TR-91-3036*, May 1991.
3. Matthews, C. D., "VUAERO Primer and Reference Manual," *WL-TR-91-3037*, Volumes I and II, 1991.
4. *OSF/Motif Style Guide Revision 1.1*, Prentice Hall, New Jersey, 1991.
5. Deters, K. J., "QUADSTREAM User's/Reference Manual," *WL-TR-92-3071*, May 1992.
6. Ely, W. L., Moore, M. E., "Stability and Control of Hypervelocity Vehicles, Volume I: Technical Report," *AFWAL-TR-88-3076*, July 1988.
7. Ames Research Staff, "Equations, Tables, and Charts for Compressible Flow," *Report 1135*, Ames Aeronautical Laboratory.
8. Lees, Lester, "Laminar Heat Transfer Over Blunt-Nose Bodies at Hypersonic Flight Speed", *Jet Propulsion*, April 1956.
9. Detra, H. W., and Hidalgo, H., "Generalized Heat Transfer Formulae and Graphs", *AVCO Research Report 72*, March 1960.
10. Cato, G. C., "Heat Transfer to the Leading Edge of a Yawed Wing", *Memorandum A-260-TH-57-115*.
11. Beckwith, I. E., and Gallagher, J. J., "Local Heat Transfer and Recovery Temperatures on a Yawed Cylinder at a Mach Number of 4.15 and High Reynolds Number", *NASA TR R-104*, 1961.

*Supersonic/Hypersonic Arbitrary Body Program*

*Mark V Release*

*Appendix A, INPUT DATA INSTRUCTIONS*

## TABLE OF CONTENTS

Section	Page
A1. INTRODUCTION .....	A-1
A2. INPUT TO MAIN EXECUTIVE ROUTINE.....	A-3
A3. GEOMETRY DATA PREPARATION .....	A-5
A3.1 Geometry Nomenclature .....	A-5
A3.2 Geometry Definition Input.....	A-6
A3.2.1 Element Reading Routine.....	A-11
A3.2.2 Elliptical-Cross-Section Data.....	A-15
A3.2.3 Parametric-Cubic Input Data .....	A-19
A3.2.4 Aircraft Geometry Option.....	A-24
A4. AERODYNAMIC PROGRAM INPUT DATA.....	A-53
A4.1 Input to Aero Executive Routine.....	A-53
A4.2 Flow Field Option Input Data .....	A-56
A4.3 Shielding Option Input Data .....	A-64
A4.4 Inviscid Pressure Option Input Data.....	A-66
A4.5 Viscous Analysis Option.....	A-73
A4.5.1 Level 1 Viscous Analysis.....	A-75
A4.5.2 Level 2 Viscous Analysis.....	A-78
A4.6 Special Routines Option Input Data.....	A-82
A4.6.1 Summation Routine Option.....	A-82
A4.6. Surface Pressure Output .....	A-84
A5. AUXILIARY PROGRAMS .....	A-85
A5.1 Stability And Control Output.....	A-85
A5.2 General Cutting Plan Option.....	A-90
APPENDIX A REFERENCES.....	A-93

## A1. INTRODUCTION

This computer program system uses a free-form approach to the preparation of the input data. That is, the order of the input cards depends upon the requirements of the problem being solved. This is true of both the system control data and the input data to each of the major program components. In the Mark V program all of the input data is read right at the beginning of the job and optionally may be printed to the output unit. In this way the user always has a record of the input data along with the output results of the program.

This program is entitled the "Supersonic-Hypersonic Arbitrary-Body Program." This title brings with it certain inherent challenges. If we are going to permit a completely arbitrary shape, we will have to use a large amount of data in describing it for the computer - we must be willing to pay something for the freedom of arbitrariness. Also, since no single pressure calculation method will give good answers for all possible vehicle shapes under all flight conditions, we must have available a large number of force calculation methods and know how and when to use them. The flow field computation capabilities permit the first order investigation of flow field interference effects between vehicle components. Here again, the user must have a good idea of what can adequately prepare the flow field generation input data. Don't expect the program to do as good a job as a complete three-dimensional method of characteristics would be able to do.

The user of this program is cautioned to closely follow the instructions given in this manual. He should not rely on his experience with the old Mark IV program as some of the input data formats have changed. However, as with any similar set of documents, no written manuals are a substitute for a complete understanding of the problem to be solved, a methodical approach to the preparation and checking of the input data, and a careful analysis of the output data. Also, the accuracy of this program in any given application depends upon the wisdom of the engineer in selecting the proper flow field and force-calculation methods.

The general scheme used in describing the input data is shown below.

Column	Variable	Routine Format	Explanation
--------	----------	-------------------	-------------

The meaning of each of these columns is as follows:

Column - Indicates the position on the card for each data field.

Variable - Gives the FORTRAN name used in the read statement by the program

Routine - Indicates the subroutine where the data are read.

Format - Indicates the FORTRAN format of the data read statement. The parameter I4 would indicate that the parameter is an integer, right justified in a field that is 4 columns wide. The parameter F10.0 would indicate a fixed point number with a decimal point and sign, and placed anywhere in the columns indicated.

Explanation - The description of the input data parameters, flags, etc.

**A2. INPUT TO MAIN EXECUTIVE ROUTINE**

Only two input cards are required for the Executive Program.

Executive Flag Card (2I1, 1X, 15A4, 1X, I1,I2)

This card must be the first card in the data deck.

Column	Variable	Routine Format	Explanation
1	IEROR	Main I1	Not Used
2	INMONT	Main I1	Not Used
3-64	TITLE	Main 15A4	Title used in the output file
65	IECHO	Main I1	0 = Do not echo input data 1 = echo input data to output file
66-67	INP_VER	Main I2	0 = Mark IV 1 = Mark V

System Control Card (20I1)

Column	Variable	Routine Format	Explanation
1	IPG(1)	Main 20I1	System options in the order that they are to be executed.
2	IPG(2)		
3	IPG(3)		= 1 Call GEOM = 2 Call AERO = 3 Call AUXILI
.etc.	.etc.		
20	IPG (20)		



### A3. GEOMETRY DATA PREPARATION

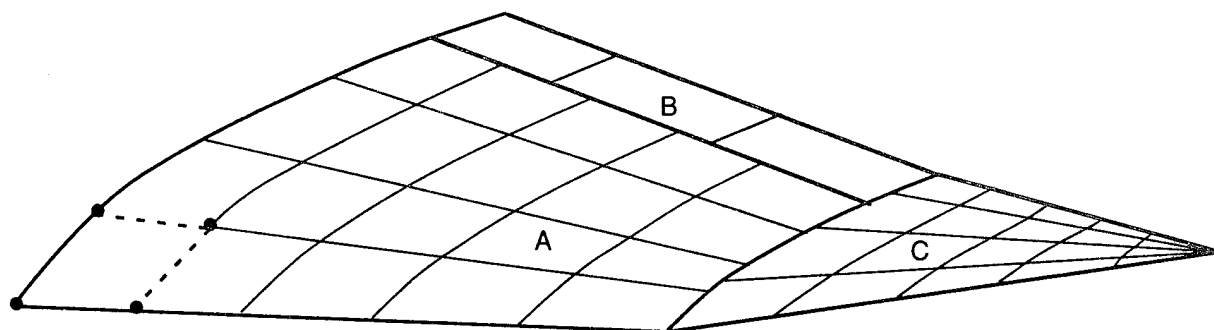
#### A3.1 Geometry Nomenclature

Before proceeding with the detailed descriptions of the geometry input data several (and often confused) geometry terms should be defined.

- Surface Element: This, the smallest geometry unit, consists of four related points on the surface of the vehicle and the area enclosed by lines connecting successive points. All geometry data must eventually be made available to the program in surface-element form.
- Plane Quadrilateral Element: Each surface element is converted by the program into a plane quadrilateral element. The plane quadrilateral element is the basic geometric unit used in the force calculations. This unit, in effect, is the integration step size and is fixed once the surface element representation of the shape is established.
- Vehicle Section: A vehicle section consists of an aggregation of surface elements that have similar size and proportions. On the Type 3 element data cards a Section starts when a point with STATUS = 2 is found. A Section ends when the next STATUS = 2 (or 3) is found.
- Vehicle Panel: A panel consists of one or more sections. On the Type 3 element data cards a Panel is ended when a STATUS = 3 is found.
- Vehicle Component: A Component is defined as a major part of the vehicle that is to be analyzed by the program as a unit (i.e., a wing, tail, etc.). A Component is usually made up of several Panels.

These various definitions are illustrated in the table below and in Figure A-1.

Basic Quantity	Combine	To Form	Examples
Points	→	Elements	
Elements	→	Sections	Inboard Section A, Flap Section B, Outboard Section C.
Sections	→	Panels	Sections A,B,C form upper wing panel.
Panels	→	Components	Upper and lower panels form wing component.
Components	→	Vehicle	Wing, fuselage, tail, etc. components form the complete vehicle.



**Figure A-1, Geometry Definitions**

The geometry data are stored on the Quadrilateral Element Storage Unit (unit 4) by PANELS. That is, each PANEL is identified by a sequence number. The PANELS are assigned consecutive sequence numbers (ISTAT3) by the GEOMetry routine as they are stored on the Quadrilateral Storage Unit. The user must keep track of the panel sequence numbers for each panel (each STATUS = 3) stored on Unit 4. The panel sequence numbers are used to group panels into components which are stored by consecutive sequence numbers and component name. These component sequence numbers are required as input to the other parts of the program to retrieve the proper geometry data for the flow field, inviscid pressure, and skin friction calculations.

### **A3.2 Geometry Definition Input**

There are two basic methods for defining the geometry in S/HABP. The first method is via Type 3 cards which contain X,Y,Z position and information on how to connect points to form elements, sections and panels. The second method is to use the geometry generation routines (Elliptical Cross Sections, Parametric Cubic Surface, and Aircraft Geometry). The second of these methods has been functionally superseded by the geometry modeler in the VECC Graphical User Interface (GUI). However the geometry generation routines of S/HABP Mark IV are still included in the S/HABP Mark V code and could be accessed by the user if run stand-alone (see Appendix A for input data instructions). Therefore the geometry generation routines input are also discussed.

Geometry Control Card (I2, 5I1, I2, 15A4, I1)

This is the first card for the GEOM option.

Column	Variable	Routine Format	Explanation
1-2	IOUT	GEOM I2	Output storage unit for Type 3 element data cards when they are generated by one of the geometry generation routines or loaded by the Element Load routine. Usually input = 8, but it may also be convenient to use it = 1 (or = 5 if INMONT = 1 on the Executive Flag Card). If quadrilaterals are not to be calculated IOUT may be input = 7 to obtain an output file of the Type 3 element cards.
3	IREW	GEOM I1	Rewind flag for unit IOUT. = 0 Rewind unit IOUT before storing any Type 3 cards on it. = 1 Do not rewind unit IOUT.
4	PRINTS	GEOM I1	Print flag for detailed quadrilateral characteristics. = 0 Do not print. = 1 Print.
5	IQUAD	GEOM I1	Quadrilateral Calculation Flag. = 0 Program will expect to use the geometry generation or storage routines to put elements on to unit IOUT. A Panel Identification card will be next in the deck.  = 1 Program will by-pass the element generation and storage options. A Panel Identification Card will not be read. Program will read Type 3 geometry cards from unit IOUT and convert them to quadrilaterals.  = 2 Same as = 0 above except that the program will not calculate the quadrilaterals. A Return from the GEOM routine will be called after the geometry generation or storage on unit IOUT is completed.

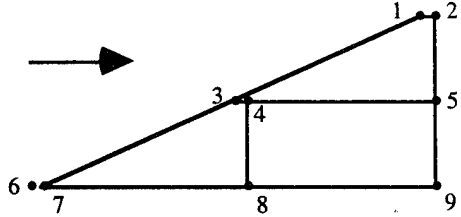
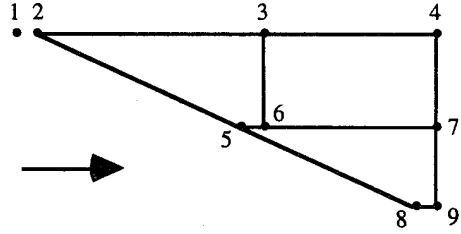
Geometry Control Card (I2, 5I1, I2, 15A4, I1) (continued)

Column	Variable	Routine Format	Explanation
6	I3MAX	GEOM I1	Number of Status 3's in the element unit IOU. Used in determining when the end of the quadrilateral computations is reached (used only when IQUAD = 1). When IQUAD $\neq$ 1 I3MAX is determined by the program.
7	NEW	GEOM I1	New data set flag. = 0 This is a new data set unit for geometry data. Set up all flags and pointers.  = 1 This is not a new data set. Use old flags and pointers to store additional panels.
8-9	NPMAX	GEOM I2	Maximum number of panels to be provided for on the new geometry data unit. Used only when NEW = 0. If input as = 0 then the program will set NPMAX = 50.
10-69	CONFIG	GEOM 15A4	Configuration identification to be written on the first record of the geometry storage unit (4).
70	IUNITS	GEOM I1	Units Flag. 0 = Inches 1 = Feet 2 = Millimeters 3 = Centimeters 4 = Meters Note: This parameter required for viscous analysis and labels in the GUI.

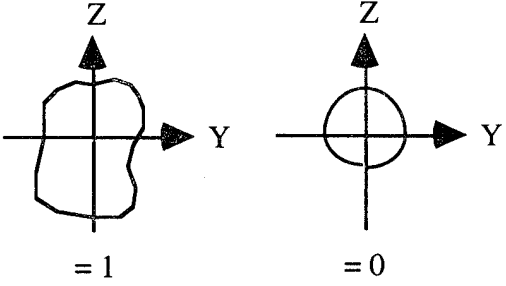
Panel Identification Card (A4,2I1,10I1,2I1,4I2,I1,2X,A6)

This card is required if IQUAD  $\neq$  1. This card is used to control the flow to the various geometry generation routines. One of these cards is required before each entry into the geometry generation DO loop. The geometry generation DO loop is terminated by the LAST parameter on this card. Each Panel Identification Card can cause the geometry generation routines to be entered up to 10 times. The number of times that the geometry generation routines are to be entered is controlled by the IGEOM(I) parameter. When IGEOM(I) = 0 the DO loop is stopped. If IGEOM(1) = 0 the DO loop will not even be started but the Panel Identification Cards will be read in until LAST = 1.

# Appendix A, Input Data Instructions

Column	Variable	Routine Format	Explanation
1-4	PANEL(I)	GEOM A4	Panel number or code identification. This identification will be stored on the quadrilateral data storage unit 4.
5	LAST	GEOM I1	Last Panel flag. = 0 This is not the last Panel Identification Card. = 1 This is the last Panel Identification Card.
6	IORN(I)	GEOM I1	<p>Geometry orientation flag (same as in the Mark III program).</p> <p>= 0 Normal mode using cross-sections.            = 1 Geometry data input in streamwise strips but for each streamwise strip of elements the first coordinate point in the right-hand strip of points is not used in the formation of the leading edge element but is ignored by the program. This is illustrated in the diagrams below for the lower wing of a vehicle. The streamwise direction is indicated by the arrow.</p>  <p>In the diagram above the numbers indicate the order of the input points. Note that points 1 and 2 are duplicate points (same values for X, Y and Z). The first element is formed by points 1-2-4-5 and point 3 is ignored. In a similar manner, an element is formed by points 3-4-7-8 and point 6 is not used.</p> <p>= 3 Same as =2 above except that the left point is ignored in the formation of the leading edge elements. This would be useful for upper surfaces of a delta wing. The input schematic for this case is shown below.</p> 

# Appendix A, Input Data Instructions

7 8 9 10 . . etc. 16	IGEOM(I)	GEOM 1011	<p>Geometry generation method flag.</p> <p>= 0 The geometry generation routines (IELE, ELLIP, CUBIC, AIRCFT) will not be called. If LAST = 0 another Panel Identification Card will be expected next.</p> <p>= 1 Routine IELE will be called to read Type 3 cards from the input unit and to copy these cards onto the geometry storage unit, IOU.</p> <p>= 2 Routine ELLIP will be called to generate elements using the ellipse generation techniques.</p> <p>= 3 Routine CUBIC will be called to generate elements using the parametric cubic method.</p> <p>= 4 Routine AIRCFT will be called to generate elements using the aircraft geometry option.</p>
17	SYMFACT(I)	GEOM 11	<p>Symmetry flag. This flag indicates the type of vehicle symmetry to be used for this component of the vehicle (see diagrams below).</p> <div style="text-align: center;">  </div> <p>Note: Although it is possible to use different Symmetry flags for different components of a vehicle, the safe thing to do is to input or generate the geometry using the same Symmetry flag for all parts of a vehicle. This is particularly important if pictures are to be drawn with a computer graphics picture drawing program.</p>
18	IFA(I)	GEOM 11	<p>Scale factor flag. This flag permits the alteration of geometry data either by a shift of the reference coordinate system or by a multiplying factor.</p> <p>= 0 Use input geometry coordinates (no scale factors will be used). The Scale Factor Card will not be input.</p> <p>= 1 Use scale factors to scale and shift the geometry data in the basic coordinate system. Scale factors and coordinate increments are applied as the geometry data are being converted into quadrilateral data. The original element data on Unit IOU are not altered.</p> <p>i.e. <math>X_{new} = X_{input} \cdot (XSC) + DELX</math></p> <p>Note: The Scale Factor Card is input after all of the Panel Identification Cards and geometry generation cards (for IELE, ELLIP, CUBIC, AIRCFT) are input, and are the last cards read before the quadrilaterals are calculated.</p>

19-20	NADJ1	GEOM I1	not used
21-22	NADJ2	GEOM I2	not used
23-24	NADJ3	GEOM I2	not used
25-26	NADJ4	GEOM I2	not used
27	VIS_TYPE	GEOM I1	Viscous/Inviscid Flag. 0= Inviscid 1= Neither 2= Viscous 3= Inviscid 4= Both
30-45	PAN_NAME	GEOM A16	Extended panel name.

### A3.2.1 Element Reading Routine

This geometry option is used to transfer element data cards (Type 3 cards) from the input unit (usually 5 or the input monitor storage unit 1) to the geometry element data storage unit (8).

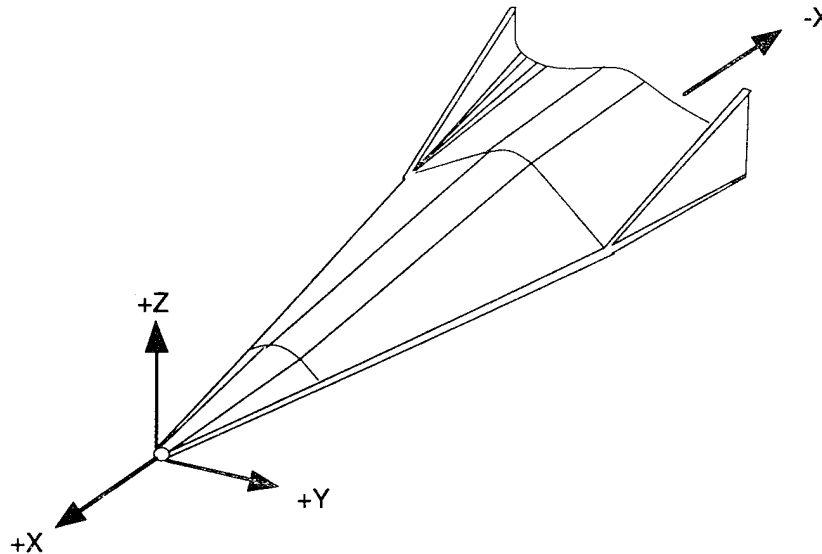
#### Element Control Card (2I2,2I1)

Column	Code	Routine Format	Explanation
1-2	I3MAX	IELE I2	The number of STATUS 3's that will be read before the IELE routine will stop and return to the main geometry program.
3-4	IN	IELE I2	Input unit for the Type 3 cards. If input = 0 the program will set IN equal to TAPEIN as defined in the main executive program (= 1 or = 5).
5	IREW	IELE I1	Rewind flag for unit IN. The rewind control statement in IELE before the geometry read starts is as follows.  IF (IN.NE.5 .AND. IREW.EQ.1) REWIND IN.  = 0 Do not rewind unit IN. = 1 If IN≠5 rewind IN.
6	I3	IELE I1	Status 3 control flag. = 0 As the element cards are copied over to unit IOUT all of the STATUS 3's will be removed except the last one.  = 1 All of the STATUS 3's will be removed as the cards are copied to unit IOUT.

Element data cards (Type 3 cards) are input following the above Element Control Card.

The important result of this general approach to the geometry problem is that the force-calculation part of the program is not affected by the method used to input the geometric shape. The form of the geometry data can be varied to meet the situation.

The coordinate system used for all the geometry data is shown in Figure A-2 below. For symmetrical vehicles it is standard practice to input the left side (+Y) of the vehicle only.



**Figure A-2 Input Geometry Coordinate System**

Since all of the geometry options finally produce geometry data in surface-element form, it is important that the methods and nomenclature used with this method be clearly understood. It is, therefore, recommended that the input instructions for the surface-element method be studied before an attempt is made to use either the ellipse or the parametric cubic options.

Under certain circumstances, the input geometry data must be input in a prescribed manner. This occurs when using the shock-expansion pressure-calculation method.

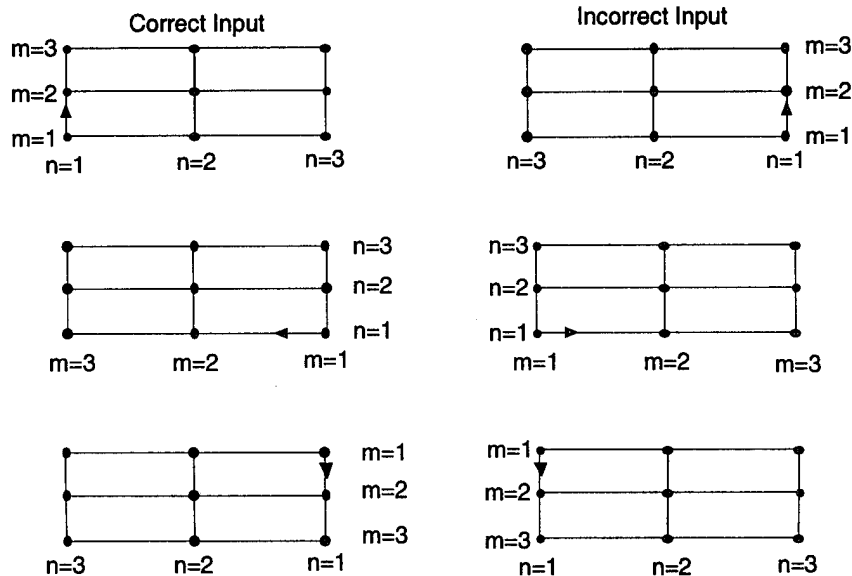
The geometric input data in this method include the coordinates of a large number of points on the vehicle surface. The input data are organized in a manner that permits the description of a vehicle on a component-buildup basis. This gives increased flexibility in shape description and makes it possible to use different force-calculation methods for different components. Because of possible changes in the surface contours of a component, it may also be necessary to divide the component into several sections. Each section of a vehicle component is further divided into a number of small units called elements, each defined by four points in space. In practice, the surface coordinates are usually recorded from cross section drawings of the vehicle in such a way that each point need be read only once (even though it may be a member of as many as four adjacent elements). Each point is defined by its three coordinates and a STATUS flag that indicates whether it is the first point of a new section, a continuation of a column of points, the beginning of a new



column, or the last point of a panel. The program uses the STATUS flags to determine how the input points are to be related to form elements, and how the elements are combined to form a section.

The first question that the user asks when starting to load the element geometry is, "In what order do I enter the surface points?" The basic rules to be followed are given below. These will be followed by a discussion of a visual technique that many users will find helpful in determining the proper loading order.

For the purpose of organizing the input data for computation, each point is assigned a pair of integers,  $m$  and  $n$ . These integers are not actually input to the program (they are calculated internally) but their use in the following discussion will provide a better understanding of the input-data organization. For each point,  $n$  identifies the "column" of points to which it belongs, and  $m$  identifies its position in the "column," i.e., the "row." The first point of a "column" always has  $m = 1$ . To ensure that the program will compute outward normal vectors, the following condition for the order of input points must be satisfied. If an observer is located in the flow and is oriented so that locally he sees points on the surface with  $m$  values increasing upward, he must also see  $n$  values increasing toward the right. Strict adherence to this simple rule will always lead to a correct set of input geometry data. Examples of correct and incorrect input are shown in Figure A-3 below. In these pictures the flow field lies above the paper, and the interior of the body lies below the paper. The arrows indicate the order of reading the points.



**Figure A-3 Correct Input Ordering**

Associated with each input point is an input quantity called its status. The first point of each new section has Status = 2. Except for the first  $n$ -line of a section, the first point of each  $n$ -line has Status 1. The last point of a panel of the vehicle has Status 3. All other points have Status = 0,

i.e., they may be left blank on the input sheet. The program will not exit properly from the surface-data subprogram and into the force-calculation phase until it reads a Status = 3.

All of the geometric input data for this geometry option are input on Type 3 Element Data cards. Each card contains the X, Y, Z coordinates and Status flag for two points on the body surface. Every card in the element-geometry deck must contain two surface points except the last card, which may have only the first-surface-point coordinates and status filled in. If a particular line of vehicle points is odd in number then it is usually advisable to repeat the last point (a dummy point) so that the last card will have two sets of point data. This permits the shifting of vehicle sections of the deck without disrupting other sections.

Element Data Input Cards (3F10.0, I1,3F10.0,I1,2X,I2,1A4,I2,4X,I4)

Column	Variable	Routine Format	Explanation
1-10	X	IELE F10.0	X-coordinate of surface point (the value of X is written anywhere in this space with a decimal point and sign; usually input only if it is negative.)
11-20	Y	IELE F10.0	Y-coordinate of surface point.
21-30	Z	IELE F10.0	Z-coordinate of surface point
31	STAT	IELE I1	Status flag for the above set of coordinates (= 2, 1, 0, or 3).
32-41	XX	IELE F10.0	X-coordinate of surface point
42-51	YY	IELE F10.0	Y-coordinate of surface point
52-61	ZZ	IELE F10.0	Z-coordinate of surface point.
62	STATT	IELE I1	Status flag for the above set of coordinates (= 2, 1, 0, or 3).
65-66	CASE	IELE I2	Case number (right-justified integer).
67-70	SECT	IELE 1A4	Numbers or letters to identify the vehicle section. These must be legal machine characters.
71-72	TYPE	IELE I2	Card type number = 03.
77-80	SEQ	IELE I4	Card sequence number. This number is used to identify each card of a particular section and to aid in keeping the cards in order (right-justified integer).

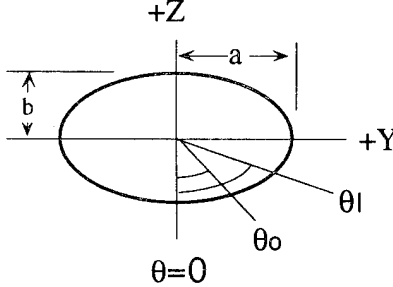
### **A3.2.2 Elliptical-Cross-Section Data**

This geometry option provides the capability of generating geometric data for vehicle components having whole or partial circular or elliptical cross sections with a minimum amount of input information required. This option is usually used to generate hemispherical noses, circular bodies, and wing and tail leading edges.

The data generated by this option are saved on the geometry storage tape (Tape 8) in normal surface-element input data form. In this manner it is possible to describe a vehicle with a combination of both hand-input data (in surface-element or parametric-cubic input form) and analytically derived circular or elliptical cross-section data.

The input data for this geometry option are described below. The input procedure is to define the basic properties of a circular or elliptical cross section (a cut in the Z-Y plane with X being a constant for the cross section). Each cross section where a set of element data are desired must be input in this manner. The first cross section must be toward the front of the vehicle, and each succeeding section must be toward the rear.

Ellipse Generation Control Card (12A4,11X,2L1,3X12,1A4,I2)

Column	Variable	Routine Format	Explanation
1-48	TITLE	ELLIP 12A4	Vehicle section or component title. Any acceptable machine characters.
60	DISCON	ELLIP I1	<p>Angular-data option flag. This flag controls the angular division of the cross section and the dummy points generated to give complete card output for the geometry storage tape. See sketch below.</p>  <p>The angular-data options are given below.</p> <p>= 1 All initial angles, <math>\theta_0</math>, and all final angles, <math>\theta_1</math> are the same for each cross section for this section of the vehicle.</p> <p>= 2 All <math>\theta_1</math> in the vehicle section are the same but the <math>\theta_0</math> varies.</p> <p>= 3 All <math>\theta_0</math> in the vehicle section are the same but the <math>\theta_1</math> varies.</p>
61	IPRINT	ELLIP I1	<p>Print flag. This flag controls the printing of the element data generated in this option. This data printout will contain the exact information written on the geometry storage tape.</p> <p>= 0 Do not print data.</p> <p>= 1 Print.</p>

Ellipse Generation Control Card (continued)

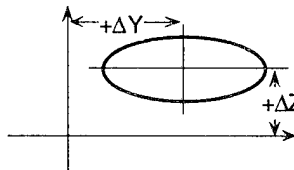
Column	Variable	Routine Format	Explanation
65-66	CASE	ELLIP I2	Case Number. A right-justified integer used to identify the vehicle data.
67-70	SECT	ELLIP 1A4	Section identification. A number or letter used to identify this section or component of the vehicle. Any acceptable machine characters.
71-72	TYPE	ELLIP I2	Card Type number = 04 (integer).

Cross-Section Data Cards (F10.0,2F6.0,I3,2F10.0,2F7.0,I1,10X12)

One card for each cross section cut desired.

Column	Variable	Routine Format	Explanation
1-10	X	ELLIP F10.0	X-station (usually negative if the vehicle nose is at the coordinate system origin).
11-16	THETO	ELLIP F6.0	Initial angle, $\theta_0$ , Degrees.
17-22	THETL	ELLIP F6.0	Final angle, $\theta_1$ Degrees.
23-25	NN	ELLIP I3	Number of divisions of cross section desired. This number controls the number and spacing of the elements generated between $\theta_0$ and $\theta_1$ Right-justified integer.
26-35	A	ELLIP F10.0	Ellipse radius along the Y- axis, a.

Cross-Section Data Cards (Continued)

Column	Variable	Routine Format	Explanation
36-45	B	ELLIP F10.0	Ellipse radius along the Z- axis, b.
46-52	DELZ	ELLIP F7.0	Offset of center of ellipse in the Z-direction, $\Delta Z$
53-59	DELY	ELLIP F7.0	Offset of center of ellipse in the Y - direction. $\Delta Y$ 
60	LAST	ELLIP I1	<p>Last Flag. This flag controls the Status flag (STATT) of the last element point generated and the position of the geometry data storage tape (Tape 8) after the element data has been written on it.</p> <p>= 0 This is not the last cross section; set STATT = 0 and read in new cross-section card.</p> <p>= 1 Not active. Do not use.</p> <p>= 2 This is the last cross section for this vehicle section or component. Set the status flag STATT = 0, and read in a new ellipse data title card.</p> <p>= 3 Not active. Do not use.</p> <p>= 4 This is the last cross section; no more sections are given, set last STATT = 3, write end of file on geometry tape.</p>
65-66	CASE	ELLIP I2	Case number (right-justified integer).
67-70	SECT	ELLIP 1A4	Numbers or letters to identify the vehicle panel. These must be legal machine characters.
71-72	ITYPE	ELLIP I2	Card type number = 5 (integer).
77-80			Card sequence number. Not read by program.

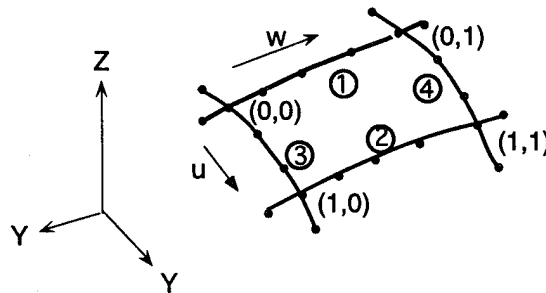
### A3.2.3 Parametric-Cubic Input Data

This geometry-input option is provided as an alternate input method in the description of arbitrary shapes. In this respect, it serves the same purpose as the surface-element input method.

In the surface-element input method, a vehicle section is described by a large number of surface points organized in an element fashion. In the Parametric Cubic method, only points along the boundaries of a patch are input to the program and the distributed surface points (surface elements) required for the subsequent quadrilateral calculations are determined by the program.

The basic features of this method are that (1) fewer input points are required to describe a shape, (2) the input of this data is a little more complicated, and (3) the generated element size is controlled by two input parameters and may be changed to meet the requirements of the problem.

The input data consist of points along the four boundaries of a patch. The program calculates the coefficients for a mathematical surface-fit equation to provide a description of the interior surface of the patch. This surface is then converted into exactly the same form as the normal surface-element input data for further calculations. The element data generated is saved on the geometry storage tape (Tape 8) for use in other phases of the program.



**Figure A-4 Parametric Cubic Patch Input.**

Each of the four boundaries is identified in Figure A-4 by a number inside a circle. The input data for each of these boundaries must be input in the order indicated by these numbers, i.e., boundaries 1, 2, 3, and 4. The order of the input points on a boundary and the order of the boundaries is important. The approach to ensure a correct input of the data is similar to that used for the quadrilateral-element input data. First, the user should imagine that he is holding a small model of the vehicle in hand. The vehicle is divided into a number of sections or patches, Figure A-6 represents one such patch. The objective here is to describe how the data for one patch is loaded into the program.

The user orients the model of the vehicle so that the number 1 boundary is to the left and the number 2 boundary to the right. Coordinates of points along the number 1 boundary are loaded first. The order of these points (from the user's view of the model) is from the bottom to the top of the patch. Note that a point must be included outside the patch at either end of the boundary to give

proper slopes at the corner points. The next input points are for boundary number 2 and again from bottom to top. Boundary number 3 is loaded from left to right as is boundary number 4. A different number of points may be used to describe each boundary up to a maximum of 20 for each one.

Each of the input points has a status flag associated with it similar to that used in the surface-element input method. The first point (the bottom point outside the patch on boundary number 1) has a status of 2. The first point on each of the other boundaries has a status of 3.

The input sheet contains two points per card. Every card must contain two points except the last which may have one point (loaded on the left side of the card).

The detailed input information required for this geometry-method option is presented below.



Parametric Cubic Title Card (12A4,1X,I3,1X,I3,3X,3I1,2XI2,1A4,I2)

This card contains patch control data and divisions to be used in converting the patch to element data.

Column	Variable	Routine Format	Explanation
1-48	TITLE	CUBIC 12A4	Section or patch title. Any acceptable machine characters.
50-52	NOU	CUBIC I3	Number of division of the parametric variable u. This controls the number of elements in the element mesh in the u-direction (right-justified integer).
54-56	NOW	CUBIC I3	Number of divisions of the parametric variable w. This controls the number of elements in the element mesh in the w-direction (right-justified integer). If this number is an even number then the program will change it to the next higher odd number so that there will always be an odd number of elements in a column. This will give an even number of points in a column to fill out both the left and right sides of the element data card.
60	LAST	CUBIC I1	<p>Last Flag. This flag controls the Status flag (STAT) of the last element point generated and the position of the geometry data storage tape (Tape 8) after all data has been written on it.</p> <p>= 1 Not active. Do not use.</p> <p>= 2 This is not the last patch. Set the last-point status flag STAT = 0, and read in a new set of patch data (including a new title card).</p> <p>= 3 Not active. Do not use.</p> <p>= 4 This is the last patch. Set the last-point status to TT = 3, write an end of file on unit IOUT.</p>
61	ISOVR	CUBIC I1	<p>First-point status override flag.</p> <p>= 0 The status flag for the first coordinate point of the patch will be = 2 (normal mode).</p> <p>= 1 The status flag for the first coordinate point of the patch will be set = 1. This will permit "joining together" several parametric cubic patches to form a single section of surface-element data.</p>

Parametric Cubic Title Card (Continued)

Column	Variable	Routine Format	Explanation
62	IPRINT	CUBIC I1	Print flag. This flag controls the printing of the element data generated in this option. This data printout will contain the exact information written on the geometry storage tape (in BCD card image form).  = 0 Do not print data.  = 1 Print.
65-66	CASE	CUBIC I2	Case number (right -justified integer).
67-70	SECT	CUBIC 1A4	Numbers or letters to identify the vehicle panel.
71-72	ITYPE	CUBIC I2	Card type number = 06.

Parametric Cubic Boundary Data (3F10.4,I1,3F10.4,I1,8X,I2)

This card contains the coordinates of the boundary curves for a parametric cubic patch.

Column	Variable	Routine Format	Explanation
1-10	X	CUBIC F10.4	X-coordinate of boundary curve point.
11-20	Y	CUBIC F10.4	Y-coordinate of boundary curve point.
21-30	Z	CUBIC F10.4	Z-coordinate of boundary curve point.
31	STAT	CUBIC I1	Status flag for the above set of coordinates (= 2, 1, 0, or 3). This flag controls the reading in of the boundary curve data and is not the same as the STATUS flag that will be generated and written on the geometry storage tape along with the generated surface element data.
32-41	XX	CUBIC F10.4	X-coordinate of boundary curve point.
42-51	YY	CUBIC F10.4	Y-coordinate of boundary curve point.
52-61	ZZ	CUBIC F10.4	Z-coordinate of boundary curve point.

Parametric Cubic Boundary Data (Continued)

Column	Variable	Routine Format	Explanation
62	STATT	CUBIC I1	Status flag for the above set of coordinates (= 2, 1, 0, or 3). This flag controls the reading in of the boundary curve data and is not the same as the STATUS flag that will be generated and written on the geometry storage tape along with the generated surface element data.
65-66	CASE		Case number. Not read by program.
67-70	SECT		Numbers or letters to identify the vehicle panel. Not read by program.
71-72	ITYPE	CUBIC I2	Card type number = 07.
77-80	SEQ		Card sequence number. Not read by program.

### A3.2.4 Aircraft Geometry Option

The Aircraft geometry Option has the capability of generating element data for six classes of aircraft surfaces. These are identified as follows.

#### 1. Wings

Airfoil ordinates are input at percent-chord locations along with the X, Y, Z coordinates of each airfoil leading edge. Wing camber data and chord lengths are also input.

#### 2. Fuselage

A fuselage may be defined in segments and may be circular or arbitrary in cross section. If the fuselage is circular it may be input by a cross-sectional area distribution. If the fuselage is arbitrary it may be input as X-Y-Z cross-sectional coordinate data. Up to four fuselage segments may be specified on each entry into the Aircraft Geometry Option.

#### 3. Pods or Nacelles

The X-Y-Z coordinates of the pod origin are input along with a pod radii distribution. Up to nine pods may be specified on each entry to the Aircraft Geometry Option.

#### 4. Fins

The X-Y-Z coordinates of the lower and upper airfoil leading edge of vertical fins are input along with airfoil ordinates at up to 10 chord locations. The chord lengths are also input. As many as six fins may be used on each entry to the Aircraft Geometry Option. Only symmetrical airfoils are generated.

#### 5. Canards or Horizontal Tails

The X-Y-Z coordinates of the inboard and outboard airfoil leading edge are input along with the airfoil ordinates and chord lengths. Up to two canards may be input on each entry to the Aircraft Geometry Option. Nonsymmetric airfoils are permitted.

#### 6. General Airfoil Surface

The orientation of each airfoil is specified by X-Y-Z coordinates of the leading and trailing edges and a rotation angle. The airfoil ordinates and camber data are also

input. This permits the description of wing or tail type of surfaces where the airfoils are not oriented in a fixed streamwise plane.

The first five classes of surfaces indicated above are the same as those available in the NASA Wave Drag program. The sixth surface is an additional feature provided within S/HABP that allows the geometric description of a surface composed of airfoil sections that may be arbitrarily oriented in space. This removes some of the restrictions imposed within the wing, fin, and canard options used in the NASA Wave Drag program. Also, additional parameters may be specified on the pod input data to allow arbitrary orientation of the pods or nacelles.

Various combinations of the above shapes may be used in describing most aircraft configurations. However, in some situations a part of a vehicle may not be accurately described by one of the above components. In this case, the particular part of the vehicle may be input or generated using the completely arbitrary shape capabilities in the other parts of the program (i.e., input elements, parametric cubic patches, etc.) For other problems it may be easiest to generate the vehicle using the Aircraft Geometry Option and then alter those cards that need changes by hand in order to give an accurate representation of the shape. This may be necessary to accurately describe such regions as wing roots, fillets, etc.

Note that in the description of each of the surfaces above certain restrictions exist as to the maximum number of fuselage segments, pods, fins, etc., that may be generated on a single entry into the Aircraft Geometry Option. It should be noted, however, that all such limitations may be overcome by entering the Aircraft Geometry Option as many times as may be required.

The output data for the Aircraft Geometry Option consist of element data cards with two coordinate points and accompanying Status flags recorded on each card (Type 3 Arbitrary-Body Program cards). These cards are written on the Geometry Storage Unit (Unit 8) for use by the rest of S/HABP. The input generated in this manner may be used directly as input data for subsequent runs on S/HABP or as input to the Douglas Arbitrary-Body Supersonic Wave Drag Program and the Douglas Potential Flow Program (the Neumann Program).

The various methods with the Aircraft Geometry Program are selected by input flags on a control card. The various parameters, tables, etc., for each aircraft component are given on a separate set of cards for each type of surface. The order and identification of each of the input cards is given in the list below. Each card, if it is to be used, must be in the order indicated.

<u>Card or Card Set Number</u>	<u>Card Identification</u>
1	Title and Identification Card
2	Control Flag Card
3	Wing Area Card (if required)
4	Wing Percent-Chord Location Card(s)
5	Airfoil Leading Edge Coordinate Card(s)
6	Wing Camber Line Card(s) (for each airfoil, if required)
7	Wing Airfoil Ordinate Card(s) (for each airfoil)
8	Fuselage X-Section Card(s) (for first segment)
9	Fuselage Camber Card(s) (if required)
10	Fuselage Cross-section Area Card(s) (if required)
11	Fuselage Y-Ordinates (for arbitrary shape) (if required)
12	Fuselage Z-Ordinates (for arbitrary shape) (if required)
13	Repeat 11 and 12 for all cross sections of segment.
14	Repeat 8 through 13 for all fuselage segments.
15	Pod Origin Card
16	Pod X-Station Card(s)
17	Pod Radii Card(s)
18	Repeat 15 through 17 for all pods.
19	Fin Leading Edge Coordinate Card
20	Fin Percent-Chord Location Card
21	Fin Airfoil Ordinate Card
22	Repeat 19 through 21 for all fins.

<u>Card or Card Set Number</u>	<u>Card Identification</u>
23	Canard Leading Edge Coordinate Card
24	Canard Percent-Chord Location Card
25	Canard Upper Ordinate Card
26	Canard Lower Ordinate Card (if required)
27	Repeat 23 through 26 for all canards
28	General Airfoil Surface Control Flag Card
29	Airfoil Percent-Chord Location Card(s)
30	Airfoil Orientation Card(s)
31	Airfoil Camber Line Card(s)
32	Airfoil Ordinate-Thickness Card(s)
33	Repeat 28 through 32 for multiple airfoil surfaces as required.
34	Repeat 1 through 33 for multiple configurations as required.
35	Type 99 card (Normal Return to Executive Program).

The detailed descriptions given on the following pages include all input cards and parameters. Where it might be useful the variable names used in the program are also given. On some of the cards identification information is located in columns 73-80. Although this information is not used by the program its use may help to eliminate errors in ordering input. The card field to be used for input numbers is indicated for each card. All integers should right justified. Real numbers may be located anywhere in the field specified.

The type 99 card contains a 99 in card columns 71 and 72. The remainder of the card has the same format as the Title and Identification Card (card 1), however these remaining fields are usually left blank.

## Aircraft Geometry Identification And Control

Title and Identification Card (9A3,8A4,I1,2X,I1,1XI2,4XA2)

Column	Variable	Routine Format	Explanation
1-59	CARD	AIRCFT 9A3, 8A4	The title that is to appear at the top of the output pages. Any acceptable machine characters.
60	ISTAT3	AIRCFT I1	Flag to control the generation of a dummy element with a very small surface area in order to introduce a Status = 3 at the appropriate time in a geometry deck. = 0 Dummy element will be included at the very end of the Type 3 cards produced in the Aircraft Geometry Option. This means that the last Type 3 card will have a Status 3 flag.  = 1 No dummy Status 3 element will be generated. The last data point in the Aircraft Geometry cards produced will have a Status = 0.
63	IHARIS	AIRCFT I1	NASA-Harris Input Coordinate Flag. = 0 Arbitrary -Body Program coordinates will be used. The usual practice is to have the nose of the vehicle at the origin of the coordinate system with the tail having a negative X-station.  = 1 The NASA Harris coordinate system will be used. The vehicle nose is at the origin and all the X-coordinates are positive for the input data on the Aircraft Geometry Data cards. The program will change them to negative values before the final Type 3 cards are written on the storage unit to be consistent with the Arbitrary-Body system.
65-66	CASE	AIRCFT I2	Case number to be printed at the top of data output pages and in card columns 65-66 on the Type 3 cards produced by the Aircraft Geometry Option.
71-72	TYPE	AIRCFT A2	Case termination flag. If input = 99 the Aircraft Geometry Option will stop with this card and return to the Geometry routine. If left blank the program will continue reading in Aircraft Geometry data cards.



Control Flag Card (7I3,I1,I2,16I3,8X)

Column	Variable	Routine Format	Explanation
1-3	JO	AIRCFT I3	Wing area flag. = 0 Wing area card is not included. = 1 Wing area card is to be read.
4-6	J1	AIRCFT I3	Wing data and camber flag. = 0 No wing data are used. = 1 Cambered wing data are to be read. = -1 Uncambered wing data are to be read.
7-9	J2	AIRCFT I3	Fuselage control flag. = 0 No fuselage data are used. = 1 Data for arbitrarily shaped fuselage will be read. = -1 Data for circular fuselage will be read.
10-12	J3	AIRCFT I3	Pod control flag. = 0 No pod data are used. = 1 Pod data are to be read.
13-15	J4	AIRCFT I3	Fin control flag. = 0 No fin data are used. = 1 Fin data are to be read.
16-18	J5	AIRCFT I3	Canard control flag. = 0 No canard data are used = 1 Canard data are to be read.
19-21	J6	AIRCFT I3	Fuselage camber and symmetry flag. = 0 Fuselage camber data will be read. = 1 Configuration is symmetrical with respect to the X-Y plane (uncambered circular fuselage is used). = -1 Fuselage is uncambered. = 2 Uncambered, arbitrary fuselage.
22	J7	AIRCFT I1	General airfoil surface flag. = 0 No airfoil data are used. = 1 General airfoil surface control card to be read.
23-24	NWAF	AIRCFT I2	Number of airfoils used to describe the wing = 2 to 20.
25-27	NWAFOR	AIRCFT I3	Number of percent-chord points used to define each wing airfoil section = 3 to 30.
28-30	NFUS	AIRCFT I3	The number of fuselage segments to be read = 1 to 4.
31-33	NRADX(1)	AIRCFT I3	Number of Y-Z coordinate points used to describe each cross section for the first fuselage segment. This parameter is used for both arbitrary and circular fuselage segments. = 3 to 30.

Control Flag Card (continued)

Column	Card	Routine Format	Explanation
34-36	NFORX(1)	AIRCFT I3	Number of X cross sections to be used for each fuselage segment. = 2 to 30..
37-39	NRADX(2)	AIRCFT I3	Same as Field 31-33 for second fuselage segment.
4-42	NFORX(2)	AIRCFT I3	Same as Field 34-36 for second fuselage segment
43-45	NRADX(3)	AIRCFT I3	Same as Field 31-33 for third fuselage segment
46-48	NFORX(3)	AIRCFT I3	Same as Field 34-36 for third fuselage segment
49-51	NRADX(A4)	AIRCFT I3	Same as Field 31-33 for fourth fuselage segment
52-54	NFORX(4)	AIRCFT I3	Same as Field 34-36 for fourth fuselage segment
55-57	NP	AIRCFT I3	Number of pods to be input (up to 9).
58-60	NPODOR	AIRCFT I3	Number of stations to be used in the pod radii distribution input. This is the same for all pods. = 2 to 30.
61-63	NF	AIRCFT I3	Number of vertical fins to be input (up to 6).
64-66	NFINOR	AIRCFT I3	Number of ordinates used to define each fin airfoil. This is the same for all fins. = 3 to 10.
67-69	NCAN	AIRCFT I3	Number of canards or horizontal tails to be input (up to 2).
70-72	NCANOR	AIRCFT I3	Number of ordinates used to define the airfoils. This is the same for all canards. = 3 to 10    Airfoil is symmetrical, upper ordinates only will be read = -3 to -10    Airfoil is unsymmetrical, lower ordinates will be read right after the upper values are read in.

Wing Area Card (F7.2)

Column	Variable	Routine Format	Explanation
1-7	FEFA	AIRCFT F7.2	This card is required if JO = 1 on the Control Flag Card. This parameter is not used by the program but may be present in some decks set up for the NASA Wave Drag Program. If JO = 0 on the Control Flag Card then the Wing Area Card is not included in the deck.

**WINGS**

The input information required by the Aircraft Geometry Option to define a wing with streamwise airfoils is as follows:

1. Number of airfoils.
2. Number of airfoil percent-chord points used to define the airfoils.
3. A table of percent-chord locations that are to be used for the airfoil thickness and camber distribution.
4. The X-Y-Z coordinates of the leading edge of each airfoil.
5. The chord length of each airfoil.
6. The airfoil ordinate data in percent of chord length at each percent-chord position for each airfoil.
7. A flag to indicate when camber data are to be read in or set equal to zero.
8. Camber values of the mean camber line ( Z ) at each percent-chord location for each airfoil.

The input information required to describe a wing is illustrated in Figure A-5.

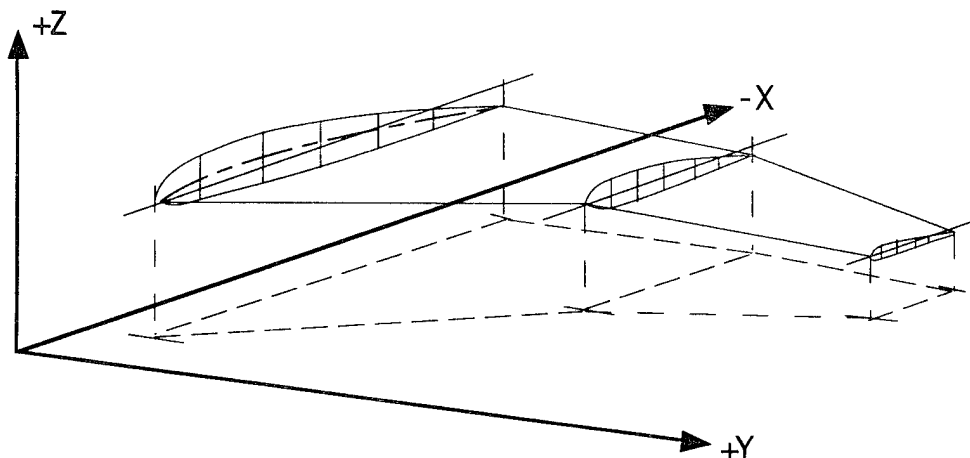


Figure A-5 Wing With Streamwise Airfoils

Note that each airfoil lies in an X-Z plane at a fixed spanwise station, Y. This will pose problems in some applications at the wing-fuselage juncture, particularly for area-ruled fuselages.

The Z-coordinates of each surface point are calculated from the following relationship.

$$Z = ZO + DZ * C * WAFORD(I,J) + TZORD(I,J)$$

where

Z = final Z-coordinate of a point on the airfoil.

DZ = upper surface - lower surface factor.  
       = +1.0 for upper surface.  
       = -1.0 for lower surface

C = chord length/100.0

WAFORD = upper airfoil thickness in percent of chord length  
           subscript I = the airfoil number (= -1 for the inboard airfoil)  
           subscript J = the number of chordwise location.

TZORD = camber, Z

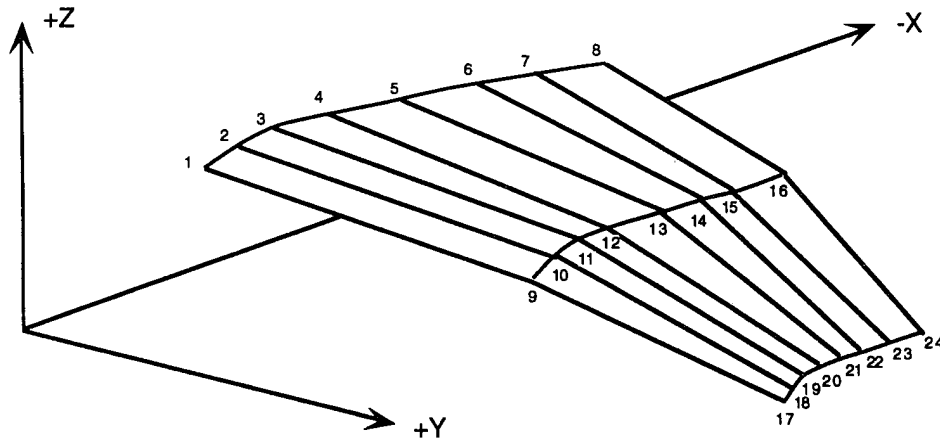
ZO = Z-coordinate of the airfoil leading edge point.

The order of the generated X-Y-Z surface points is shown in Figure A-6(a) for the upper surface of the wing. The wing lower surface is shown in Figure A-6(b).

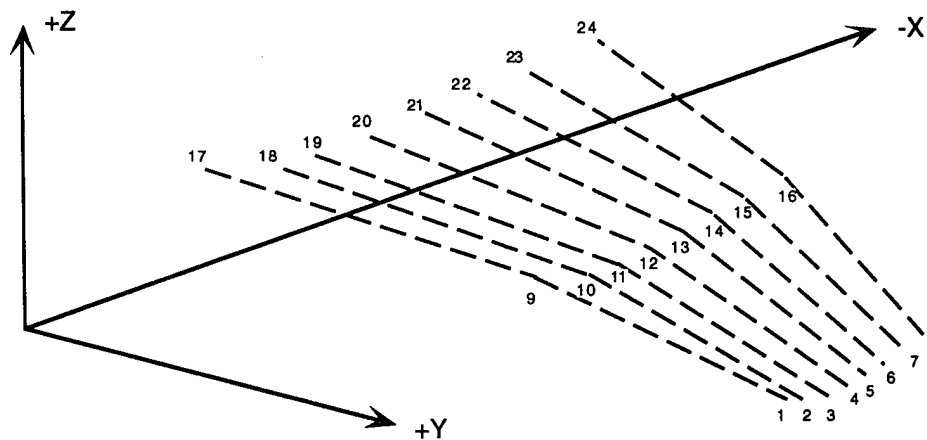
In the wing shown in Figure A-6 there were 7 percent-chord locations from the leading edge to the trailing edge. Since each element data card generated (Type 3 card) contains two data points, three and one-half cards will be required for the root upper surface. Rather than beginning the next wing chord on the last half of the third card, this field is filled by a dummy point (point 8) which is a repeat of the trailing edge point (point 7). This dummy point is furnished automatically by the

program when it is required and permits each airfoil to be started on a new card. This facilitates the manual manipulation of resulting data decks to meet various needs.

The wing lower surface is considered as a new vehicle section. To obtain the correct "outward" side of the surface the generation of points starts at the tip rather than the root as was done for the upper surface. Dummy trailing edge points are generated just as was done for the upper surface. In the example shown, point 1 will have a Status flag of 2, points 9 and 17 will have Status = 1, and all the rest will have Status - 0.



(a) Upper Surface



(b) Lower Surface

**Figure A-6 Points And Elements Generated For Wing Surface**

Note that a surface point is calculated at each input percent-chord location for each airfoil. The number of percent-chord locations and the number of airfoils will determine the number of surface

points generated and the number of resulting surface elements. In the example shown in Figure 9 a total of 28 elements were generated (including 4 that result from the dummy trailing edge points).

### WING DATA CARDS

#### Wing Percent-Chord Location Card(s) (10F7.0,10X)

Column	Variable	Routine Format	Explanation
1-7	XAF(1)	AIRCFT 10F7.0	Table of percent-chord locations that are to be used for the airfoil thickness and camber coordinates. Use as many cards as required with 10 numbers on each card. Use as many fields and cards as is specified by NWAFOR (Field 25-27) on the Control Flag Card.
8-14	XAF(2)		
15-21	XAF(3)		
22-28	XAF(4)		
29-35	XAF(5)		
36-42	XAF(6)		
43-49	XAF(7)		
etc.	etc.		

#### Airfoil Leading Edge Coordinate Cards (4F7.0,52X)

Column	Variable	Routine Format	Explanation
1-7	WAFORG(I,1)	AIRCFT 4F7.0	X-coordinate of the airfoil leading edge.
8-14	WAFORG(I,2)		Y-coordinate of the airfoil leading edge.
15-21	WAFORG(I,3)		Z-coordinate of the airfoil leading edge.
22-28	WAGORG(I,4)		The airfoil streamwise chord length.
73-80.			May be punched WAFORG <sub>i</sub> , where I denotes the airfoil number.

Note: Repeat this card for all airfoils, starting with the inboard airfoil and working to the outboard tip airfoil. The number of these cards is given by the parameter NWAF (Field 23-24) on the Control flag Card and must not be greater than 20.

#### Wing Camber Line Cards (10F.0,10X)

Not required if J1 = -1 on the Control Flag Card.

Column	Variable	Routine Format	Explanation
1-7	TZORD(J,1)	AIRCFT 10F7.0	Camber values of the mean camber line ( Z ) at each percent-chord location for each airfoil. Use as many cards as required with 10 numbers on each card. Each airfoil must have as many numbers as was specified by the parameter NWAFOR in field 25-27 on the Control Flag Card. There will be as many sets of these cards as was indicated by the parameter NWAF in field 23-24 on the Control Flag Card. The first number for each airfoil should start on a new card. The identification TZORD <sub>j</sub> may be punched in card columns 73-80, where j denotes the airfoil number.
8-14	TZORD(J,2)		
15-21	TZORD(J,3)		
22-28	TZORD(J,4)		
29-35	TZORD(J,5)		
36-42	TZORD(J,6)		
43-49	TZORD(J,7)		
50-56	TZORD(J,8)		
57-63	TZORD(J,9)		
64-70	TZORD(J,10)		
1-7	TZORD(J,11)		
etc.	etc.		

Wing Airfoil Ordinate Cards (10F7.0,10X)

Column	Variable	Routine Format	Explanation
1-7	WAFORD(J,1)	AIRCFT 10F7.0	Wing airfoil thickness ordinates as a percent of chord length at each percent-chord ordinate position for each airfoil. Use as many cards as required with 10 numbers on each card. Each airfoil must have as many numbers as was specified by the parameter NWAFOR in field 25-27 on the Control Flag Card. There will be as many sets of these cards as was indicated by the parameter NWAF in field 23-24 of the Control Flag Card. The first number for each airfoil should start on a new card. The identification WAFORDj may be punched in card columns 73-80, where j denotes the airfoil number.
8-14	WAFORD(J,2)		
15-21	WAFORD(J,3)		
22-28	WAFORD(J,4)		
29-35	WAFORD(J,5)		
36-42	WAFORD(J,6)		
43-49	WAFORD(J,7)		
50-56	WAFORD(J,8)		
57-63	WAFORD(J,9)		
64-70	WAFORD(J,10)		
1-7	WAFORD(J,11)		
etc.	etc.		

## FUSELAGE

The input information required by the Aircraft Geometry Option to define a fuselage is as follows:

1. Fuselage shape flags (circular, arbitrary, cambered).
2. Number of fuselage segments (1 to 4).
3. Number of Y-Z coordinate points used to describe an X-cross section for each fuselage segment (3 to 30).
4. Number of X cross sections to be used for each fuselage segment (2 to 30).
5. A table of X-values of the fuselage cross sections for each fuselage segment.
6. Tables of Y-Z values to describe each cross section for arbitrary shaped fuselage.
7. Fuselage centerline camber distribution.
8. Cross-sectional area distribution of the fuselage if it is circular.

From the above input items we see that the fuselage may be circular or arbitrary in cross section, may have camber, and may be made up of as many as four segments. However, a single fuselage cannot be made up to a combination of circular and arbitrary cross sections. (This comment only applied for a single pass into the Aircraft Geometry Option. Multiple entries into the Aircraft Geometry Option from the Hypersonic Arbitrary-Body executive main program permits an unlimited combination of program capabilities.)

The order of the generated fuselage coordinate points is from the bottom around to the top. The first point for each fuselage segment has a Status flag of 2, each new cross section starts with a Status of 1, and all the other points have Status = 0. If the last point of a cross-section fills only the left half of the type 3 element data card, a dummy point is generated to fill the right field of the card. Figure A-7 shows the order of the generated surface points for an arbitrarily shaped fuselage. Only two fuselage segments are shown.



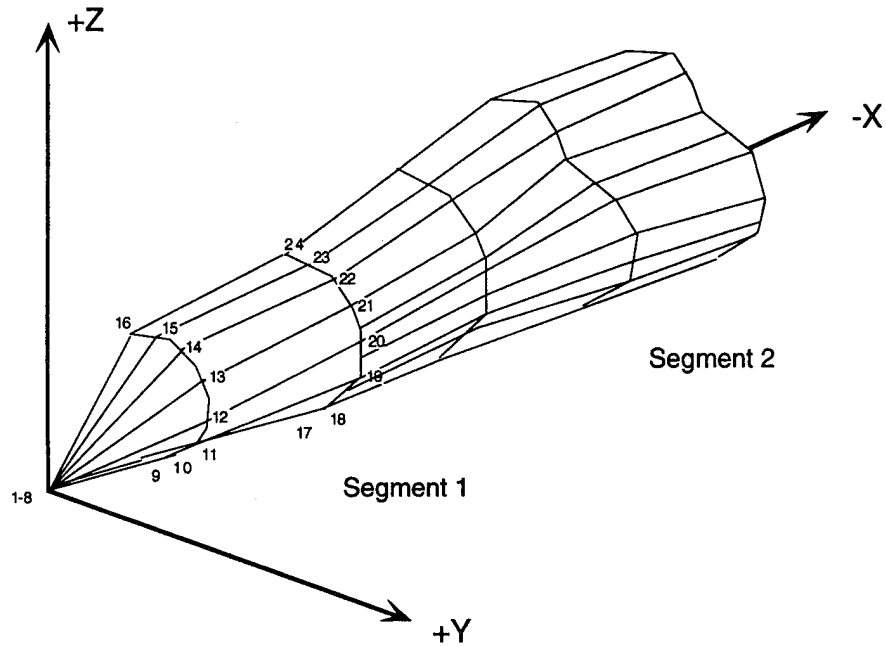


Figure A-7 Fuselage Geometry

Fuselage X-Station Card(s) (10F7.0,10X)

Column	Variable	Routine Format	Explanation
1-7	XFUS(I,1)	AIRCFT 10F7.0	Table of X-station values to be used for first fuselage segment. Use as many cards as required with 10 numbers on each card. The number of cross sections used must be the same as indicated by the parameter NFORX on the Control Flag Card. The X-stations must be in an order proceeding from the front of the vehicle to the rear.
8-14	XFUS(I,2)		
15-21	XFUS(I,3)		
22-28	XFUS(I,4)		
29-35	XFUS(I,5)		
36-42	XFUS(I,6)		
43-49	XFUS(I,7)		
50-56	XFUS(I,8)		
57-63	XFUS(I,9)		
64-70	XFUS(I,10)		
1-7	XFUS(I,11)		The identification XFUSj may be punched in card columns 73-80, where j denotes the number of the last fuselage station given on that card.
etc.	etc.		

Fuselage Chamber Card(s) (10F7.0,10X)

Required only in J6 = 0 on the Control Flag Card

Column	Variable	Routine Format	Explanation
1-7 8-14 15-21 22-28 29-35 36-42 43-49 50-56 57-63 64-70 1-7 etc.	ZFUS(I,1) ZFUS(I,2) ZFUS(I,3) ZFUS(I,4) ZFUS(I,5) ZFUS(I,6) ZFUS(I,7) ZFUS(I,8) ZFUS(I,9) ZFUS(I,10) ZFUS(I,11) etc.	AIRCFT 10F7.0	Fuselage camber distribution for first fuselage segment. Use as many cards as required with 10 numbers on each card. The number of camber points used must be the same as indicated by the parameter NFORX on the Control Flag Card. For an arbitrarily shaped fuselage this parameter will not actually be used in generating the surface coordinate points. However, if the parameter J6 = 0 on the Control Flag Card, then the appropriate number of fuselage camber cards must be present in the deck (all the values may be = 0.0).  The identification ZFUSj may be punched in card column 73-80, where j denotes the number of the last fuselage station given on the card.

Fuselage Cross-Section Area Card(s) (10F7.0,10X)

Not required if the fuselage is arbitrary in shape (if J2 = 1)

Column	Variable	Routine Format	Explanation
1-7 8-14 15-21 22-28 29-35 etc.	FUSARD(I,1) FUSARD (I,2) FUSARD (I,3) FUSARD (I,4) FUSARD (I,5) etc.	AIRCFT 10F7.0	A table of fuselage cross-sectional area distribution at each station for the first fuselage segment. Use as many cards as required with 10 numbers on each card. The number of points used must be as indicated by the parameter NFORX on the Control Flag Card. The identification FUSARDj may be punched in card columns 73-80, where j denotes the station number.

Fuselage Y-Ordinate Card(s) (10F7.0,10X)

Used for arbitrarily shaped fuselage only. Do not use these cards unless J2 = 1 on the Control Flag Card.

Column	Variable	Routine Format	Explanation
1-7	YFUSY(I,J,1)	AIRCFT 10F7.0	Y-ordinates for one fuselage cross section, starting at the bottom and going around to the top of the section. Use as many cards as required with 10 numbers on each card. The parameter NRADX on the Control Flag Card specifies the number of Y-ordinates required for each cross section. Each set of Y-ordinate cards are followed immediately by a Z-ordinate set of cards for that same cross section. The number of ordinates may range from 3 to 30.
8-14	YFUSY(I,J,2)		
15-21	YFUSY(I,J,3)		
22-28	YFUSY(I,J,4)		
29-35	YFUSY(I,J,5)		
36-42	YFUSY(I,J,6)		
43-49	YFUSY(I,J,7)		
etc.	etc.		

Fuselage Z-Ordinate (Card(s)) (10F7.0,10X)

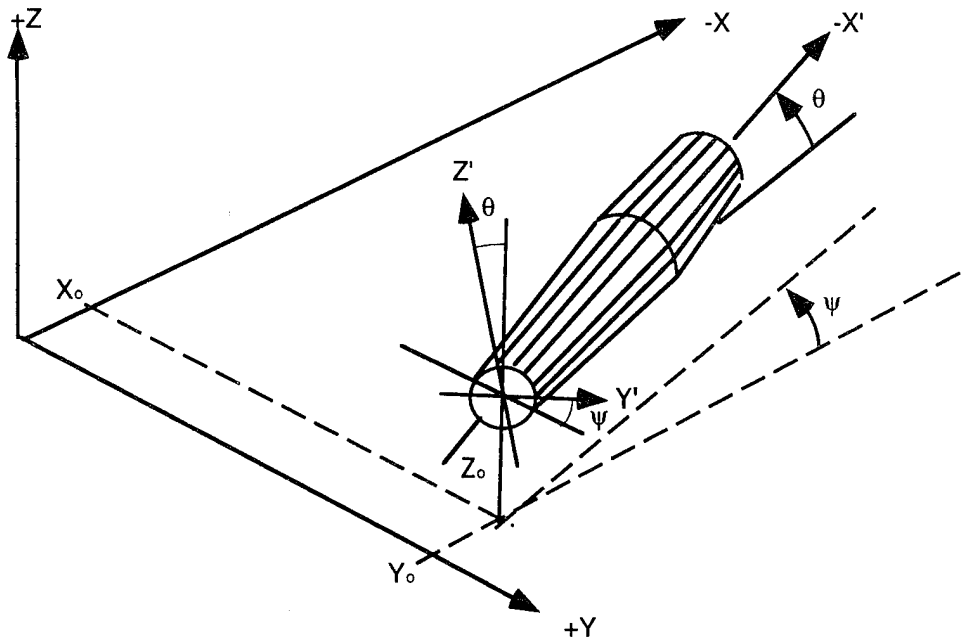
Used for arbitrarily shaped fuselage only. Do not use these cards unless J2 = 1 on the Control Flag Card.

Column	Variable	Routine Format	Explanation
1-7	ZFUSZ(I,J,1)	AIRCFT 10F7.0	Z-ordinates for one fuselage cross section, starting at the bottom and going around to the top of the section. These Z-ordinates correspond to the Y-ordinates input on the Y-ordinate card described above. Use as many cards as required with 10 numbers on each card. The parameter NRADX on the Control Flag Card specifies the number of Z-ordinates required for each cross section. Each set of Z-ordinate cards must be right behind the corresponding set of Y-ordinate cards.
8-14	ZFUSZ(I,J,2)		
15-21	ZFUSZ(I,J,3)		
22-28	ZFUSZ(I,J,4)		
29-35	ZFUSZ(I,J,5)		
36-42	ZFUSZ(I,J,6)		
43-49	ZFUSZ(I,J,7)		
50-56	ZFUSZ(I,H,8)		
57-63	ZFUSZ(I,j)		
etc.	etc.		

Note: Paired sets of Y-ordinate and Z-ordinate cards are repeated for each fuselage cross section until all cross sections for a single fuselage segment are read in. The number of paired sets is given by the parameter NFORX input on the Control Flag Card. The order of cards for the second fuselage segment is the same as prepared for the first segment. This includes fuselage station, camber, cross-section area, and Y-Z ordinates for each fuselage segment.

## PODS OR NACELLES

A pod or nacelle is a body of revolution with its axis arbitrarily located with reference to the vehicle axis system. This increased capability has been added without effecting the NASA Wave Drag Program input format (the NASA program is limited to having the pod axis parallel to the vehicle X-axis). The pod is defined with respect to its own coordinate system ( $X'-Y'-Z'$ ), the orientation of which is considered to have been achieved through a yaw-pitch sequence of rotations. The parameters used in defining the pod and the formation of surface elements are illustrated in Figure A-8.



**Figure A-8 Pod Or Nacelle Geometry**

The input information required to define a pod or nacelle is as follows:

1. Number of pods (up to 9).
2. Number of stations to be used in the pod radii distribution input (2 to 30). This is the same for all pods.
3. The X-Y-Z coordinates of the origin and end of each pod in the vehicle coordinate system.
4. A table of X-ordinates (relative to pod origin) for the pod radii distribution.
5. Pod radii distribution for each pod.

The order of the generated surface points is from the bottom around to the top. The first point of each pod has a Status of 2, each new station starts with a Status of 1, and all other points have Status = 0. If the last point for a station fills only the left half of the Type 3 Element Data Card, a dummy point is generated to fill the right half of the card. When the pod axis lies in the X-Z origin plane, only half the pod is generated ( $-90^\circ \leq w \leq +90^\circ$ ).

Otherwise elements for the complete pod are determined.

In addition to specifying the axis orientation, the number of elements in  $180^\circ$  may also be specified. If this expanded capability is not used and the input fields are left blank, the program assumes the pod axis is parallel to the vehicle axis, and elements are generated every  $15^\circ$  in  $\omega$ .

Pod Origin Card (3F7.0,3X,I1,3X,3F7.0,.6X,I1,5X,I2)

Column	Variable	Routine Format	Explanation
1-7	PODORG(I,1)	F7.0	X-coordinate of the origin of the first pod with respect to the vehicle coordinate system origin.
8-14	PODORG(I,2)	F7.0	Y-coordinate of the origin of the first pod with respect to the vehicle coordinate system origin.
15-21	PODORG(I,3)	F7.0	Z-coordinate of the origin of the first pod with respect to the vehicle coordinate system origin.
25	IOR	I1	Arbitrary orientation flag. If this value does not equal 1, fields 29-49 are ignored and the pod axis is assumed parallel to the vehicle X-axis.
29-35	PODORG(I,4)	F7.0	X-coordinate of the end point of the first pod with respect to the vehicle coordinate system.
36-42	PODORG(I,5)	F7.0	Y-coordinate of the end point of the first pod with respect to the vehicle coordinate system.
43-49	PODORG(I,6)	F7.0	Z-coordinate of the end point of the first pod with respect to the vehicle coordinate system.
56	IEL	I1	Element number flag. If this value does not equal 1, field 62-63 is ignored and 12 elements are assumed in $180^\circ$ .
62-63	NEL	I2	Number of elements in $180^\circ$ ( $\leq 36$ ).
73-80			The card identification, PODRGi, may be punched in these columns where i denotes the pod number.

## Appendix A, Input Data Instructions

### Pod X-Station Card(s) (10F7.0,10X)

Column	Variable	Routine Format	Explanation
1-7	XPOD(I,1)	AIRCFT 10F7.0	Table of X-ordinates (relative to pod origin) to be used for the pod radii distribution. Use as many cards as required with 10 numbers on each card. The number of cross sections used must be the same as indicated by the parameter NPODOR given in field 58-60 on the Control Flag Card. The first X-ordinate must be zero, and the last X-ordinate is the length of the pod. The identification XPODi may be punched in card columns 73-80, where <i>i</i> denotes the pod number.
8-14	XPOD(I,2)		
15-21	XPOD(I,3)		
22-28	XPOD(I,4)		
29-35	XPOD(I,5)		
36-42	XPOD(I,6)		
43-49	XPOD(I,7)		
50-56	XPOD(I,8)		
57-63	XPOD(I,9)		
64-70	XPOD(I,10)		
1-7	XPOD(I,11)		
etc.	etc.		

### Pod Radii Card(s) (10F7.0,10X)

Column	Variable	Routine Format	Explanation
1-7	PODR(I,1)	AIRCFT 10F7.0	A table of pod radii distribution at each X-station for the first pod. Use as many cards as required with 10 numbers on each card. The number of points used must be the same as indicated by the parameter ANPODOR given in field 58-60 on the Control Flag Card. The identification PODRi may be punched in card columns 73-80, where <i>i</i> denotes the pod number.
8-14	PODR(I,2)		
15-21	PODR(I,3)		
22-28	PODR(I,4)		
29-35	PODR(I,5)		
36-42	PODR(I,6)		
43-49	PODR(I,7)		
etc.	etc.		

Note: A new set of all three pod input card sets is required for each pod (a maximum of 9 are provided for). Note that every pod uses the same value for the parameter NPODOR. If the Y-ordinate of the pod origin is 0.0 then only half of the symmetrical centerline pod is generated. If the Y-ordinate is not equal to 0.0 then the entire pod will be generated.

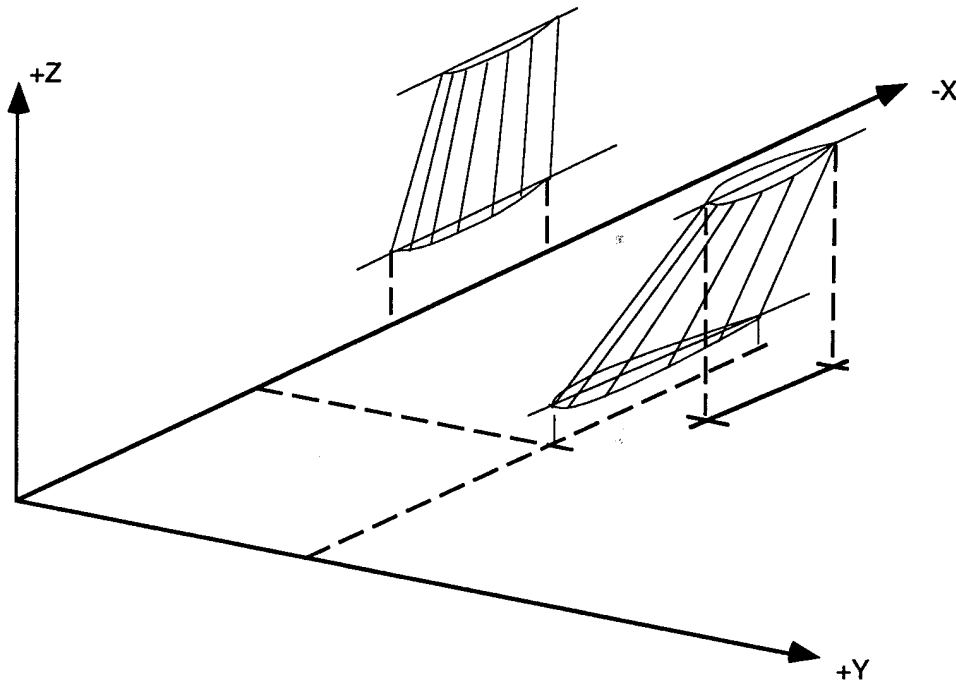
## FINS

Fins are defined by upper and lower uncambered airfoils. Each airfoil lies in an X-Y plane at a fixed vertical distance, Z. This restriction may pose problems in some applications at the fin fuselage juncture. The leading edge of each airfoil is located relative to the coordinate system origin by input X-Y-Z displacements. For fins located on the plane of symmetry only half of the surface is generated. If the fin is located off the plane of symmetry both sides will be generated. The generation of surface points and elements follows the same general procedure as outlined previously for the wing.

The input information required to define a fin is as follows:

1. Number of vertical fins (up to 6).
2. Number of ordinates used to define each fin airfoil section (3 to 10).
3. The X-Y-Z coordinates of the fin lower and upper leading edges.
4. Chord lengths for the lower and upper airfoils.
5. A table of percent-chord locations that are used to define airfoils.
6. A table of airfoil ordinates as a percent of the chord length.

The input and formation of fin surface elements are illustrated in Figure A-9.



**Figure A-9 Fin Geometry Generation**

## Appendix A, Input Data Instructions

### Fin Leading Edge Coordinate Card (8F7.0,24X)

Column	Variable	Routine Format	Explanation
1-7	FINORG(N,1,1)	AIRCFT 8F7.0	X-coordinate of root airfoil leading edge.
8-14	FINORG(N,1,2)		Y-coordinate of root airfoil leading edge.
15-21	FINORG(N,1,3)		Z-coordinate of root airfoil leading edge.
22-28	FINORG(N,1,4)		Chord length of root airfoil.
29-35	FINORG(N,2,1)		X-coordinate of tip airfoil leading edge.
36-42	FINORG(N,2,2)		Y-coordinate of tip airfoil leading edge.
43-49	FINORG(N,2,3)		Z-coordinate of tip airfoil leading edge.
50-56	FINORG(N,2,4)		Chord length of tip airfoil.
73-80			The card identification FINORGn, where n denotes the fin number.

### Fin Percent-Chord Location Card (10F.0,10X)

Column	Variable	Routine Format	Explanation
1-7	XFIN(N,1)	AIRCFT 10F7.0	Table of percent-chord locations that are to be used for the airfoil thickness ordinates. Only one card is used and may contain up to 10 numbers. The number of percent-chord locations used must be the same as indicated by the parameter NFINOR in field 64-66 on the Control Flag Card. The card identification XFINn may be punched in card columns 73-80, where n denotes the fin number.
8-14	XFIN(N,2)		
15-21	XFIN(N,3)		
22-28	XFIN(N,4)		
29-35	XFIN(N,5)		
36-42	XFIN(N,6)		
43-49	XFIN(N,7)		
50-56	XFIN(N,8)		
57-63	XFIN(N,9)		
64-70	XFIN(N,10)		

### Fin Airfoil Ordinate Card (10F7.0,10X)

Column	Variable	Routine Format	Explanation
1-7	FINORD(N,1)	AIRCFT 10F7.0	Table of fin airfoil thickness ordinates as a percent of chord length at each percent-chord ordinate position for the fin. Only one card is used and may contain up to 10 numbers. The number of percent-chord locations used must be the same as indicated by the parameter NFINOR in field 64-66 on the Control Flag Card. The card identification FINORDn may be punched in card columns 73-80, where n denotes the fin number.
8-14	FINORD(N,2)		
15-21	FINORD(N,3)		
22-28	FINORD(N,4)		
29-35	FINORD(N,5)		
36-42	FINORD(N,6)		
43-49	FINORD(N,7)		
50-56	FINORD(N,8)		
57-63	FINORD(N,9)		
64-70	FINORD(N,10)		

Note: A new set of all three fin input cards is required for each fin (a maximum of 6 are provided for). Note that every fin uses the same value for the parameter NFINOR. If the Y-ordinates of the fin leading edge are input as 0.0 only half of the symmetrical centerline fin is generated. If the Y-ordinates of the leading edge are not equal to 0.0 then the entire fin will be generated.



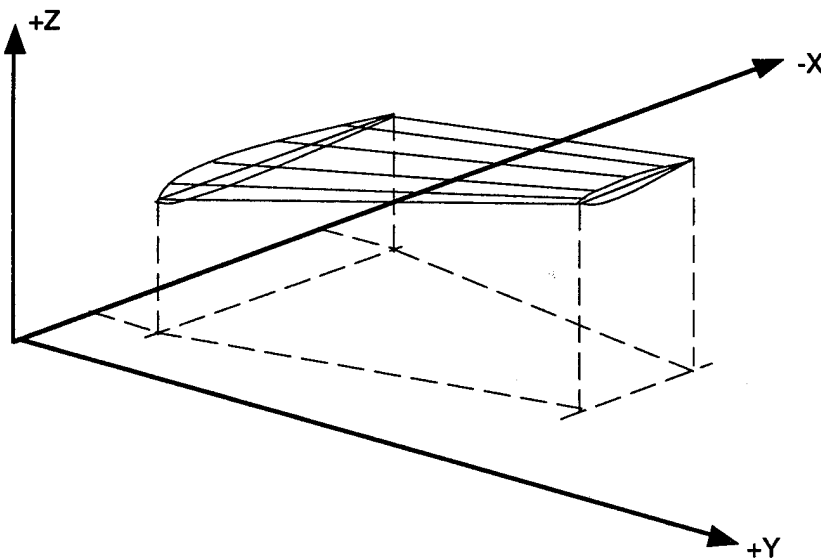
### CANARDS OR HORIZONTAL TAILS

Canards or horizontal tails are defined in a manner similar to that used for fins using an inboard airfoil and an outboard airfoil. Each airfoil lies in an X-Z plane at a fixed Y distance. The airfoils may be symmetrical or unsymmetrical. Both the top and bottom of the surface will be generated using the same procedures as outlined for the wing.

The input information required to define a canard or horizontal tail is as follows:

1. Number of canards (up to 2).
2. Number of ordinates used to define the canard airfoils (3 to 10).
3. The X-Y-Z coordinates of the inboard and outboard airfoil leading edges.
4. Chord lengths of the inboard and outboard airfoils.
5. A table of percent-chord locations that are to be used to define the airfoils.
6. A table of airfoil upper surface ordinates as a percent of chord length. If the airfoil is not symmetrical another table contains the lower surface ordinates.

The generation of canard and horizontal tail surfaces is illustrated in Figure A-10.



**Figure A-10 Generation Of Canard and Horizontal Tail Geometry**

Canard Leading Edge Coordinate Card (8F7.0,24X)

Column	Variable	Routine Format	Explanation
1-7 8-14 15-21 22-28	CANORG(N,1,1) CANORG(N,1,2) CANORG(N,1,3) CANORG(N,1,4)	AIRCFT 8F7.0	X-coordinate of inboard airfoil leading edge. Y-coordinate of inboard airfoil leading edge. Z-coordinate of inboard airfoil leading edge. Chord length of inboard airfoil.
29-35 36-42 43-49 50-56	CANORG(N,2,1) CANORG(N,2,2) CANORG(N,2,3) CANORG(N,2,4)		X-coordinate of outboard airfoil leading edge. Y-coordinate of outboard airfoil leading edge. Z-coordinate of outboard airfoil leading edge. Chord length of outboard airfoil.
73-80			The card identification CANORGn may be punched in card columns 73-80 where n denotes the fin number.

Canard Percent-Chord Location Card (10F7.0,10X)

Column	Variable	Routine Format	Explanation
1-7 8-14 15-21 22-28 29-35 36-42 43-49 50-56 57-63 64-70	XCAN(N,1) XCAN(N,2) XCAN(N,3) XCAN(N,4) XCAN(N,5) XCAN(N,6) XCAN(N,7) XCAN(N,8) XCAN(N,9) XCAN(N,10)	AIRCFT 10F7.0	Table of percent-chord locations that are to be used for the airfoil thickness ordinates. Only one card is used and may contain up to 10 numbers. The number of percent-chord locations used must be the same as indicated by the parameter NCANOR in field 70-72 on the Control Flag Card. The card identification XCANn may be punched in card columns 73-80, where n denotes the canard number.

Canard Upper Ordinate Card (10F7.0,10X)

Also used for lower surface if canard is symmetrical.

Column	Variable	Routine Format	Explanation
1-7 8-14 15-21 22-28 29-35 36-42 43-49 50-56 57-63 64-70	CANORD(N,1) CANORD(N,2) CANORD(N,3) CANORD(N,4) CANORD(N,5) CANORD(N,6) CANORD(N,7) CANORD(N,8) CANORD(N,9) CANORD(N,10)	AIRCFT 10F7.0	Table of canard airfoil thickness ordinates as a percent of chord length at each percent-chord ordinate position. Only one card is used and may contain up to 10 numbers. The number of percent-chord locations used must be the same as indicated by the parameter NCANOR in field 70-72 on the Control Flag Card. If the parameter NCANOR is positive (+) these airfoil ordinates will be used for both the top and bottom of the canard. The card identification CANORDn may be punched in card columns 73-80, where n denotes the canard number.  If the canard airfoil is not symmetrical the bottom airfoil thickness ordinates are input of the following card.

Canard Lower Ordinate Card (10F7.0,10X)

This card is only used if the airfoil is not symmetrical ( $\text{NCANOR} < 0$ )

Column	Variable	Routine Format	Explanation
1-7	CANOR1(N,1)	AIRCFT 10F7.0	Table of canard airfoil thickness ordinates for the lower surface as a percent of chord length at each percent-chord ordinate position. Only one card is used and may contain up to 10 numbers. The number of percent-chord locations used must be the same as indicated by the parameter NCANOR in field 70-72 on the Control Flag Card. The parameter NCANOR must be negative. Both the upper and lower thickness ordinates are input as positive percent-of-chord values. The card identification CANOR1n may be punched in card columns 73-80, where n denotes the canard number.
8-14	CANRO1(N,2)		
15-21	CANOR1(N,3)		
22-28	CANOR1(N,4)		
29-35	CANOR1(N,5)		
36-42	CANOR1(N,6)		
43-49	CANOR1(N,7)		
50-56	CANOR1(N,8)		
57-63	CANOR1(N,9)		
64-70	CANOR1(N,10)		

Note: A new set of all canard cards is required for each canard (a maximum of 2 canards are provided for). Note that every canard uses the same value for the parameter NCANOR.

### GENERAL AIRFOIL SURFACES

This geometry surface type may be used to generate surfaces that are defined by airfoil sections having arbitrary orientations in space. The airfoils are not confined to fixed planes as was the case for the wings, fins and canards previously described. This more general approach permits the use of non-streamwise airfoil sections and is useful in describing intersecting components such as the wing and tail fuselage junctures. Input cards for this surface type cannot be used as input to the NASA Wave Drag Program.

The general airfoil surface is defined by connecting two or more airfoil sections with straight lines. The orientation of each airfoil is given by coordinates of the leading and trailing edges and an airfoil rotation angle. The techniques used in defining these airfoils and in performing the necessary transformation to obtain the required Z-Y-Z coordinates in the vehicle coordinate system are discussed below.

Each airfoil section is defined relative to a coordinate system fixed within the airfoil. The airfoil thickness displacements may be measured either from the mean-camber line along a line perpendicular to the airfoil axis or on a line that is normal to the mean camber line. This latter method is used in some of the early NASA airfoil documents. All airfoil section parameters are expressed as a percent of the airfoil chord. The parameters used in defining an airfoil are illustrated in Figure A-11. In this illustration the airfoil lies in the  $\eta$ - $\zeta$  plane.

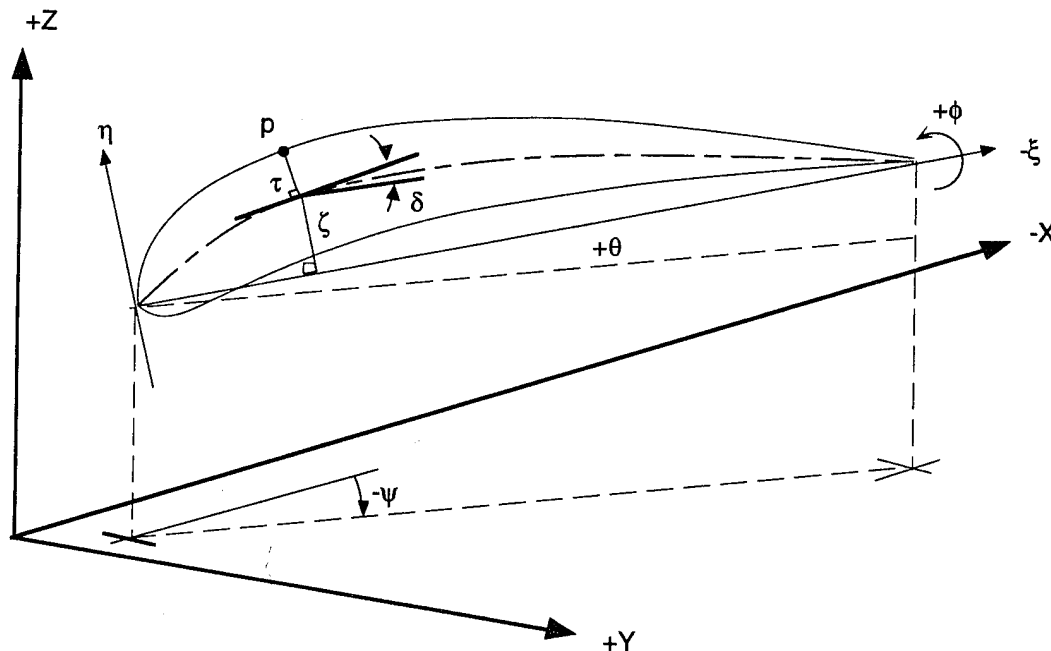


Figure A-11 General Airfoil Coordinate System

The input information required by the Aircraft Geometry Option to define a general airfoil surface is as follows:

1. Number of airfoils.
2. Number of airfoil percent-chord points used to define the airfoils.
3. Flags to control the thickness distribution type, generation of tip and root closure elements, and repetitive use of mean camber line and thickness distributions.
4. A table of percent chord locations that are to be used for the airfoil thickness and camber distributions.
5. The X-Y-Z coordinates of the leading and trailing edge of each airfoil section.
6. The roll angle  $\phi$  of each airfoil section.
7. The mean camber line ordinates in percent-chord at each percent chord location for each airfoil.
8. Thickness distribution in percent chord at each percent-chord position for each airfoil.

The roll angle  $\phi$  is input explicate and together with  $\psi$  and  $\theta$  are positive in the right-handed sense of the reference system.

Zero values for the rotation angles indicate the airfoil is oriented parallel to the X-Z plane. Zero yaw and pitch angles and a +90 degree roll angle gives an airfoil in the X-Y plane (such as a vertical tail root airfoil).

This surface type differs from those previously described in that repetitive use may be made of the arbitrary airfoil option on a single pass into the Aircraft Geometry Option. This stacking option allows wings, fins, etc., to be generated on a single pass into the Aircraft Geometry Option. A control flag also permits repetitive use of airfoil data for subsequent airfoils to save input time when all the surface airfoils are identical. Tip and root closure elements may also be generated to give a completely enclosed surface.

The arrangement of the generated X-Y-Z surface points and elements is similar to the procedure outlined for the wing surface with the exception of elements that may be generated to close the tip and root sections.

General Airfoil Control Flag Card (22X,I3, 2(3X,I2), 7(4X,I1),2X,A4)

Column	Variable	Routine Format	Explanation
1-22			Surface description statement. Any acceptable machine characters. Not used by the program
23-25	ISURF	AIRCFT I3	Control surface deflection flag (=0)
29-30	NAF	AIRCFT I2	Number of airfoils used to describe the surface. =2 to 20
34-35	NAFORD	AIRCFT I2	Number of percent-chord points used to define each airfoil section. = 3 to 30
40	NCAM	AIRCFT I1	Flag for mean camber line distribution. = 0 No camber data will be input = 1 Camber data will be input
45	NACA	AIRCFT I1	Airfoil thickness type flag =0 Thickness will be calculated normal to chord line =1 Thickness will be calculated normal to the mean camber line. =2 Same as = 0, but camber input as $\Delta Z$
50	IT1P	AIRCFT I1	Flag for closure surface at the tip. =0 Do not generate tip closure surface =1 Tip closure surface will be generated.
55	IROOT	AIRCFT I1	Flag for closure at the root =0 Do not generate tip closure surface =1 Tip closure surface will be generated.
60	ISIMC	AIRCFT I1	Flag for similar camber line distribution = 0 Each airfoil camber line will be input = 1 Camber line distribution will be the same for all airfoils and need be input only for the first.
65	ISIMT	AIRCFT I1	Flag for similar thickness distribution. = 0 Each airfoil thickness distribution will be input. = 1 Thickness distribution will be the same for all airfoils and need be input only for the first.
70	MORE	AIRCFT I1	Flag to indicate stacking of general airfoil surfaces. = 0 This is the last surface. = 1 Another surface follows and a complete set of arbitrary airfoil input cards is expected immediately following this set.
73-76	SURFID	AIRCFT A4	Surface identification to be punched in fields 73-76 of the output element data Type 3 cards. Any acceptable machine characters.

Airfoil Percent-Chord Location Card(s) (10F7.0)

Column	Variable	Routine Format	Explanation
1-7 8-14 15-21 22-28 29-35 36-42 43-49 50-56 57-63 64-70	XOC(1) XOC(2) XOC(3) XOC(4) XOC(5) XOC(6) XOC(7) XOC(8) XOC(9) XOC(10)	AIRCFT 10F7.0	Table of percent-chord locations that are to be used for the airfoil thickness and camber distributions. Each card may contain up to 10 fields, if more are required continue with additional cards of the same format. Use as many fields as is specified by ANAAFORDF (field 34-35) on the General Airfoil Control Flag Card. Maximum number of fields is 30 (3 cards).
1-7 etc.	XOV(11) etc.		Fields 73-80 of each card may be used for identification.

Airfoil Orientation Cards (10F7.0)

Column	Variable	Routine Format	Explanation
1-7 8-14 15-21 22-28 29-35 36-42 43-49	AFORG(I,1) AFORG(I,2) AFORG(I,3) AFORG(I,4) AFORG(I,5) AFORG(I,6) AFORG(I,7)	AIRCFT 10F7.0	X-coordinate of the airfoil leading edge. Y-coordinate of the airfoil leading edge. Z-coordinate of the airfoil leading edge. X-coordinate of the airfoil trailing edge. Y-coordinate of the airfoil trailing edge. Z-coordinate of the airfoil trailing edge. Airfoil roll angle in degrees.
73-80			May be used for identification

Note: Repeat this card for all airfoils, starting with the inboard root airfoil and working to the outboard tip airfoil. The number of these cards is given by the value of NAF (field 29-30) of the General Airfoil Control Flag Card and must not be greater than 20.

Airfoil Camber Distribution Cards (10F7.0)

Required only if NCAM = 1 on the General Airfoil Control Flag Card. If NACA = 2, use wing camber cards (see page A-34).

Column	Variable	Routine Format	Explanation
1-7 8-14 15-21 22-28 29-35 36-42 43-49 50-56 57-63 64-70	AFCAM(I,1) AFCAM(I,2) AFCAM(I,3) AFCAM(I,4) AFCAM(I,5) AFCAM(I,6) AFCAM(I,7) AFCAM(I,8) AFCAM(I,9) AFCAM(I,10)	AIRCFT 10F7.0	Mean camber line distribution, in percent-chord, at each percent-chord location XOC. Use as many cards as required with 10 numbers on each card. Each airfoil must have as many numbers as was specified by the parameter NAFORD (field 34-35) on the General Airfoil Control Flag Card (30 maximum). There will be as many sets of cards as given by the parameter NAF (field 29-30) on the General Airfoil Control Flag Card. The first value for each airfoil should start on a new card. If parameter ISIMC = 1 (field 60 of General Airfoil Control Flag Card), only one set of cards is required. Fields 73-80 may be used for identification.
1-7 etc.	AFCAM(I,11) etc.		

Airfoil Thickness Distribution Cards (10F7.0)

Column	Variable	Routine Format	Explanation
1-7 8-14 15-21 22-28 29-35 36-42 43-49 50-56 57-63 64-70	AFORD(I,2) AFORD(I,2) AFORD(I,3) AFORD(I,4) AFORD(I,5) AFORD(I,6) AFORD(I,7) AFORD(I,8) AFORD(I,9) AFORD(I,10)	AIRCFT 10F7.0	Airfoil thickness coordinates, in percent-chord, at each percent-chord location XOC. Use as many cards as required with 10 numbers on each card. Each airfoil must have as many numbers as was specified by the parameter NAFORD (field 34-35) on the General Airfoil Control Flag Card (30 maximum). There will be as many sets of cards as given by the parameter NAF (field 29-30) on the General Airfoil Control Flag Card. The first value for each airfoil should start on a new card. If parameter ISIMT = 1 (field 6 of General Airfoil Control Flag Card), only one set of cards is required. Fields 73-80 may be used for identification. Note: Data are input as 1/2 of total thickness values (i.e., if $t/c_{total} = 0.04$ , then input $AFORD = t/c_{total} \cdot 2.0\%$ ).
1-7 etc.	AFORD((I,11) etc.		



## A4. AERODYNAMIC PROGRAM INPUT DATA

The Aerodynamic portion of the program contains five major components.

1. Flow Field Analysis
2. Shielding Analysis
3. Inviscid Pressures
4. Viscous Methods
5. Special Routines

Access to these major options is obtained by use of the Aero option on the System Control Card input to the Main Executive Routine. All five of the above major options are controlled by an executive routine called AERO.

The input data to the AERO executive routine includes a set of flags (IPG) to determine the sequence of calls to the above five options. Up to 20 calls may be made to the various options on a single entry into the AERO executive routine. After the last nonzero option is executed, the program will return to the Main Executive program. The basic flight conditions, vehicle reference dimensions, and the angle of attack and yaw angle cards are input to the AERO executive routine and apply for all of the AERO options called by the IPG commands.

### A4.1 Input to Aero Executive Routine

#### Aero System Title Card (15A4)

Column	Variable	Routine Format	Explanation
1-60	TITLE	AERO 15A4	Title to be printed at the top of each page of the output.

#### Aero Flag Card (21I1)

Column	Variable	Routine Format	Explanation
1 2 3 . . . etc. 20	IPG (1) IPG (2) IPG (3) . . . etc. IPG (20)	AERO I1	Aerodynamic sub-options to be used in the order which they will be solved. Maximum of 20.  = 1 Flow Field Analysis = 2 Shielding Analysis = 3 Inviscid Pressures = 4 Viscous Forces = 5 Special Routines
21	IPAERO	AERO I1	= 0 Print Summation Data Only = 1 Print Component Data Also

# Appendix A, Input Data Instructions

## Flight Condition Card (4F10.0, I1, I2, I1, 1X, F10.0)

Column	Variable	Routine Format	Explanation
1-10	MACH	AERO F10.0	Free-stream Mach number
11-20	ALT	AERO F10.0	Flight altitude (feet). If input as less than -1000 (e.g., -2000.0) free-stream pressure is input in place of PSTAG (cc 21-30) and free-stream temperature is input in place of TSTAG (cc 31-40).
21-30	PSTAG	AERO F10.0	Wind-tunnel stagnation pressure (atmospheres). = 0.0 If the U.S. 1962 Standard Atmospheric properties are to be used at the input altitude. ≠ 0.0 Input altitude will be ignored and the input stagnation pressure and temperature will be used to calculate tunnel free-stream properties (using isentropic ideal-gas relationships). When ALT < -1000, freestream pressure is input in lbs-ft <sup>2</sup> (PSF)
31-40	TSTAG	AERO F10.0	Wind-tunnel stagnation temperature, °F. This number will be used with the above pressure to calculate the tunnel free-stream properties.
41	IGAS	AERO I1	Gas selection flag. = 0 Air properties will be used. = 1 Helium properties will be used.
42-43	NAB	AERO I2	Number of α-β cards to be read. A maximum of 20 cards are permitted.
44	IDERIV	AERO I1	Aerodynamic derivatives, static & dynamic = 0 No Derivatives = 1 Pitch Static Stability Derivatives = 2 Pitch Control Derivatives = 3 Pitch Dynamic Derivatives = 4 Lat/Dir Static Stability Derivatives = 5 Lat/Dir Control Derivatives = 6 Lat/Dir Dynamic Stability Derivatives = 7 IDERIV=1,2 and 3 = 8 IDERIV=4,5 and 6 = 9 IDERIV=1-6
46-55	DANGLE	AERO F10.0	Increment used for Angular Derivative Calculations (IDERIV = 1,2,4 or 5)

## Reference Dimension Card (6F10.0)

Column	Variable	Routine Format	Explanation
1-10	SREF	AERO F10.0	Reference area for the force coefficients (wing area). Must be in units consistent with input scaled geometry data.
11-20	MAC	AERO F10.0	Reference length to be used in pitching moment calculations.
21-30	SPAN	AERO F10.0	Reference length to be used in rolling-and-yawing-moment calculations.
31-40	XCG	AERO F10.0	Longitudinal position of center of gravity for moment calculations. Note that XCG will frequently be input as a negative number since the negative X-axis is usually taken as directed from the nose to the tail.
41-50	YCG	AERO F10.0	Lateral position of the center of gravity. Usually = 0.0.
51-60	ZCG	AERO F10.0	Vertical position of the center of gravity.

$\alpha$ - $\beta$  Cards (7F10.0)

The number of these cards to be input is controlled by the parameter NAB. The complete set of  $\alpha$ - $\beta$  cards is assumed to be used by all the AERO options (Flow Field, Inviscid Pressures, Viscous, and Special) unless the various options specify otherwise.

Column	Variable	Routine Format	Explanation
1-10	ALPHA (I)	AERO F10.0	Vehicle angle of attack ( $\alpha$ ), deg.
11-20	BETA (I)	AERO F10.0	Vehicle sideslip angle ( $\beta$ ). Positive with the wind striking the right side of the vehicle, deg.
21-30	ROL (I)	AERO F10.0	Vehicle roll angle, deg. Positive with right wing down.
31-40	CDELTA(I)	AERO F10.0	NOT USED (Input CDEL on Component Organization Card).
41-50	QI (I)	AERO F10.0	Vehicle pitch rate, radians/sec.
51-60	RI (I)	AERO F10.0	Vehicle yaw rate, radians/sec.
61-70	PI (I)	AERO F10.0	Vehicle roll rate, radians/sec.

Note: The above parameters are stored in data arrays. A maximum of 20 conditions are permitted.

### A4.2 Flow Field Option Input Data

The Flow Field program is reached by way of the suboption calls from the Aero Executive routine. The flow field program is used to load or generate local flow fields for use in the pressure calculations of the Inviscid Force option. The flow field program stores local flow field data on the flow field data storage unit using direct access file techniques. The unit contains several directory tables to provide the necessary pointers to each level of the data. These include a Master Directory, a flight condition directory table, a component (region) directory table and an  $\alpha$ - $\beta$  directory table. The general format in which the flowfield information is generated and stored is summarized by the diagram in Figure A-12.

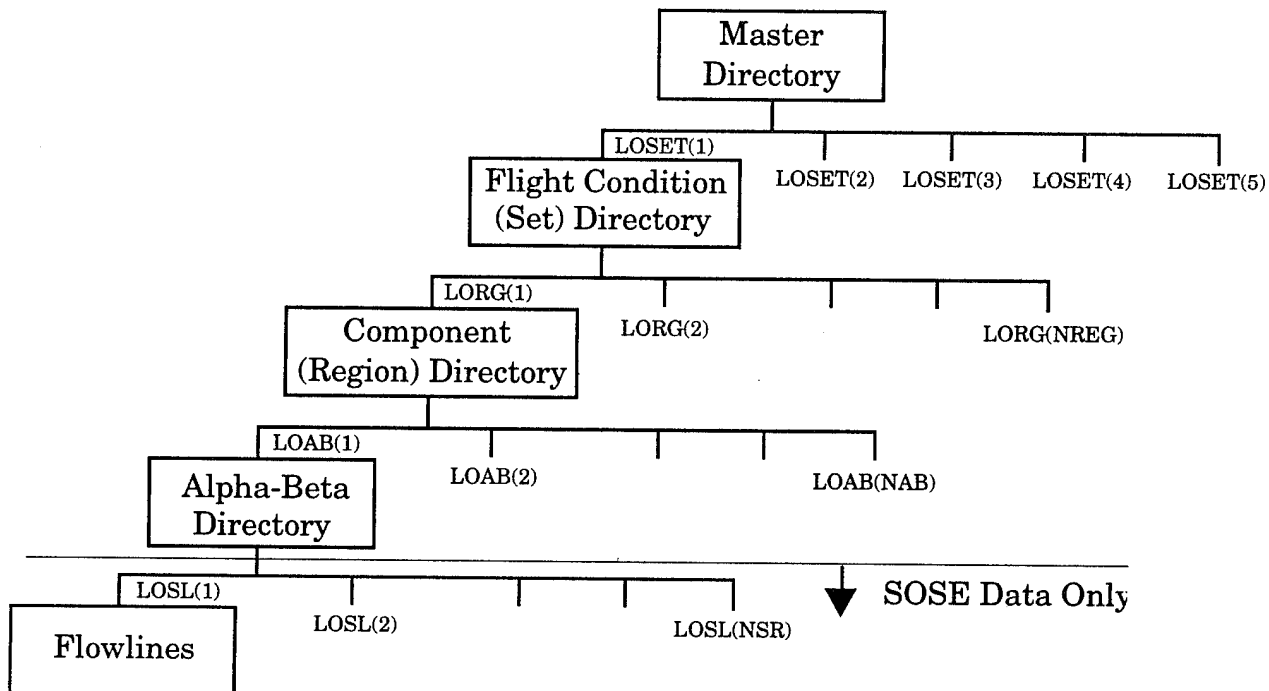


Figure A-12 Format For Storing Flowfield Information

#### Master Directory Card (I1,19x,A40)

Column	Variable	Routine Format	Explanation
1	MFLAG	FLOW I1	Master Directory Status Flag = 0 Set Up New Directory = 1 Use Existing Directory Table
2-3	NPAN	FLOW I2	Number of components (regions) to be analyzed. A maximum of FIVE components can be stored on Unit 10. Note: A component/region can consist of several panels.
21-60	TITLEM	FLOW A40	Master Directory Title

**Flowfield Analysis:** The following cards pertain directly to setting up the inputs for a flowfield analysis. The Flowfield Analysis Card replaces the Region Directory Table Card. Several additions and deletions to the original card have been made. One notable change is the addition of the component ID input, which is used to define flowlines for the current region. Other additions to this card include inputs to customize the output of the shock shape. These inputs allow the user to specify the number of points and the spacing of the points along the shock. Two options are given to space the points; (1) Even distribution, (2) Half Cosine distribution. The half cosine distribution gives a denser distribution towards the front and is recommended for blunt nose vehicles.

**Flowfield Analysis Card (I3,4I1,I3)**

Column	Variable	Routine Format	Explanation
1-3	ICOMP	FLOW I3	Component ID (For flowlines defined by streamlines, the component ID must be the same as that used in QUADSTREAM. For flowlines defined by cutting planes, the component specified must contained all panels needed to determine the cutting planes.)
4	IDTYP(1)	FLOW I1	Flowfield Analysis Type = 0 Surface data will be hand input. = 1 Surface data generated using Shock Expansion. = 2 Flowfield data will be hand input. = 3 Flowfield data will be generated using Simple Flowfield Methods. = 4 Flowfield data will be generated using Shock Expansion.
5	IPRINT	FLOW I1	Detail Print Flag = 0 Don't Print Detailed Flowfield Data = 1 Print Detailed Flowfield Data
6	IVUAERO	FLOW I1	VUAERO Print Flag = 0 Don't Print VUAERO Results File = 1 Print VUAERO Results File (Only input if IDTYP(1) = 4)
7	ISPACE	FLOW I1	Shock Output Data Spacing = 0 Even Spacing = 1 Half Cosine Spacing (Only input if IDTYP(1) = 4 and IVUAERO = 1)
8-10	NSHKPTS	FLOW I3	Number of shock points to write shock shape to VUAERO Results File (Only input if IDTYP(1) = 4 and IVUAERO = 1)

Shock Expansion Inputs:

This option is provided as a means of generating flowfield data for subsequent use in the inviscid pressure calculation part of the program. The shock expansion option can be used to generate either flowfield data for use in calculating interference effects or surface pressures for calculating forces and moments. The results of the shock expansion calculations in terms of the local surface flow properties (Mach, velocity direction cosines,  $P/P_\infty$ , and  $T/T_\infty$ ) are then stored on unit 10 using mass storage techniques.

The shock expansion inputs have not changed significantly with the exception of adding inputs to use a blunt body starting solution and read streamlines. The blunt body starting solution requires that the distance aft of the nose tip to the end of the bluntness be entered in addition to the radius of the body at that point. This information is used to calculate an equivalent spherical nose radius. The other main addition to the shock expansion inputs is the capability to read streamlines. When using this option, a streamline file must exist with the alpha-beta combinations defined on the Flight Condition Card.

Shock Expansion Flow Field Control Card (5I1,5x,2F10.0)

Column	Variable	Routine Format	Explanation
1	IORDER	FFBODY I1	Order of Shock Expansion Analysis = 1 First Order Shock Expansion. = 2 Second Order Shock Expansion.
2	IFLOW	FFBODY I1	Flowline Option = 0 Use Existing Cutting Plane Data. = 1 Generate Cutting Plane Data. = 2 Read Streamline File.
3	ISTART	FFBODY I1	Starting Solution Type = 0 Tangent Wedge or Prandtl-Meyer Starting Conditions (2-D). = 1 Tangent Cone (ARC CP792). = 2 Tangent Cone (Jones Methods). = 3 Spherical Nose Starting Solution. = 4 Circular Cylinder Starting Solution (2-D).
4	ISLOPE	FFBODY I1	Body Slope Flag = 0 Use Linear Slope Calculation. = 1 Use Circular-Arc Calculation.
5	IPRNT	FFBODY I1	Detail Print Flag = 0 Do Not Print Detail Data. = 1 Print Detail Shock Expansion Data.
11-20	RADN	FFBODY F10.0	Radius of Body at XNOSE (scaled coordinates) Used in computation of sphere radius *Only Used If ISTART > 2
21-30	XNOSE	FFBODY F10.0	Distance Aft of Nose Tip to Nose-Body Juncture (scaled coordinates). NOTE: Always > 0. *Only Used If ISTART > 2

Cutting Plane Control Card (I2,2I1)

Input only if IFLOW = 1

Column	Variable	Routine Format	Explanation
1-2	NPL	MERID I2	Number of cutting planes (maximum = 36).
3	INPHI	MERID I1	Cutting Plane Type Flag = 0 Meridian cuts, equally spaced. = 1 Meridian cuts, position input. = 2 Parallel cuts, equally spaced. = 3 Parallel cuts, position input.
4	INORM	MERID I1	Surface Normal Flag = 0 Approximate surface normal using cutting plane orientation. = 1 Use local element normal.
5	IPRT	MERID I1	Detail Print Flag = 0 Do Not Print Detailed Data. = 1 Print Detailed Cutting Plane Data.

Cutting Plane Origin and Orientation Card (6F10.0)

Input only if IFLOW = 1

Column	Variable	Routine Format	Explanation
1-10	XP0	MERID F10.0	X0 (scaled coordinates)
11-20	YP0	MERID F10.0	Y0
21-30	ZP0	MERID F10.0	Z0
31-40	PSI0	MERID F10.0	PSI0, $\Psi_0$ , degrees
41-50	THET0	MERID F10.0	THET0, $\theta_0$ , degrees
51-60	PHI0	MERID F10.0	PHI0, $\phi_0$ , degrees

Meridian Planes Input Position Cards (6F10.0)

Input only if IFLOW = 1

This card is input only if INPHI = 1. Input 6 points per card. The number of points is equal to NPL.

Column	Variable	Routine Format	Explanation
1-10	PHI (1)	MERID 6F10.0	Meridian angles in degrees. Must be in ascending order.
11-20	PHI (2)		
21-30	PHI (3)		
31-40	PHI (4)		
41-50	PHI (5)		
51-60	PHI (6)		

NOTE: Use more cards until PHI (NPL) is reached.

Parallel Planes PHI Card (1F10.0)

This card is input only if INPHI = 2 or 3, and IFLOW = 1

Column	Variable	Routine Format	Explanation
1-10	PHICD	MERID F10.0	Plane angle in degrees. ( = 0.0 for parallel streamwise cuts on a wing).

Parallel Planes Input Position Cards (3F10.0)

These cards are input only if IFLOW = 1, and INPHI = 2 or 3. If INPHI = 2 then two of these cards will be read. If INPHI = 3 then the number of these cards must be equal to the parameter NPL input on the Cutting Plane Control Card.

Column	Variable	Routine Format	Explanation
1-10	XN	MERID	The coordinates in the reference coordinate system (body coordinates) through which the cutting plane will pass.
11-20	YN	3F10.0	
21-30	ZN		

Hand Load Flowfield Data: These cards are used to directly load flowfield data on the flowfield storage unit. These cards are input only when IDTYP(1) = 2. The Hand Load Flowfield Data cards have been modified to ease the entry of experimental data. Uniform flowfield inputs have not changed. Non-Uniform flowfield data can now be loaded in any order without concern about sub-regions. The new cards that will be used to enter input for the Hand Load Flowfield Data Options are shown below.

Hand Load Data Type Card (I1)

Column	Variable	Routine Format	Explanation
1	IDTYP(2)	FFINPT I1	Flowfield Data Type Flag = 1 Uniform Flowfield data = 2 Non-Uniform Flowfield Data

Hand Load Uniform Flowfield Data Card (6F10.0)

Enter when IDTYP(2)=1

Column	Variable	Routine Format	Explanation
1-10	DINF(1)	FFINPT F10.0	Local Mach Number ( $M_{local}$ )
11-20	DINF(2)	FFINPT F10.0	X direction cosine component of local velocity vector ( $V_x/V$ )
21-30	DINF(3)	FFINPT F10.0	Y direction cosine component of local velocity vector ( $V_y/V$ )
31-40	DINF(4)	FFINPT F10.0	Z direction cosine component of local velocity vector ( $V_z/V$ )
41-50	DINF(5)	FFINPT F10.0	Local Pressure Ratio ( $P/P_\infty$ ) <sub>local</sub>
51-60	DINF(6)	FFINPT F10.0	Local Temperature Ratio ( $T/T_\infty$ ) <sub>local</sub>



Hand Load Non-Uniform Flowfield Control Card (I3)

Enter when IDTYP(2)=2

Column	Variable	Routine Format	Explanation
1-3	NPTS	FFINPT I3	Number of flowfield data points to be loaded

Hand Load Non-Uniform Flowfield Coordinate Card (3F10.0)

Column	Variable	Routine Format	Explanation
1-10	DATA(1)	FFINPT F10.0	X coordinate of flowfield data point
11-20	DATA(2)	FFINPT F10.0	Y coordinate of flowfield data point
21-30	DATA(3)	FFINPT F10.0	Z coordinate of flowfield data point

Hand Load Non-Uniform Flowfield Data Card (6F10.0)

Column	Variable	Routine Format	Explanation
1-10	DATA(4)	FFINPT F10.0	Local Mach Number ( $M_{local}$ )
11-20	DATA(5)	FFINPT F10.0	X direction cosine component of local velocity vector ( $V_x/V$ )
21-30	DATA(6)	FFINPT F10.0	Y direction cosine component of local velocity vector ( $V_y/V$ )
31-40	DATA(7)	FFINPT F10.0	Z direction cosine component of local velocity vector ( $V_z/V$ )
41-50	DATA(8)	FFINPT F10.0	Local Pressure Ratio ( $P/P_\infty$ ) <sub>local</sub>
51-60	DATA(9)	FFINPT F10.0	Local Temperature Ratio ( $T/T_\infty$ ) <sub>local</sub>

NOTE: Flowfield Coordinate Card and Data Card are input in an alternating fashion such that the program expects a Coordinate Card then a Data Card. These cards are input NPTS times.

Simple Flowfield Methods: This option is provided as a means of generating uniform flowfield data using the compression, Prandtl-Meyer expansion, cone, and Newtonian Prandtl-Meyer routines provided in the program. The basic input requirements include the freestream Mach number, the direction cosine component of the flowfield, and the flow turning angle. The program then uses one of the above routines to calculate the local Mach number,  $P/P_\infty$ , and  $T/T_\infty$  values. These data are then stored on the flowfield storage unit. Each entry to the Simple Flowfield Option generates a flowfield region. The cards below are input only if IDTYP(1) = 3. The Simple Flowfield cards have not changed from the Mark IV version of S/HABP.

Simple Flowfield Data Source Card (I1)

Column	Variable	Routine Format	Explanation
1	ISORCE	FFSPEC I1	Method Selection Flag = 1 Wedge Compression = 2 Prandtl-Meyer expansion from freestream Mach number = 3 Cone Surface Flowfield = 4 Newtonian/Prandtl-Meyer Flowfield

Simple Flowfield Data Card (6F10.0)

Column	Variable	Routine Format	Explanation
1-10	DINF(1)	FFSPEC F10.0	Freestream Mach Number
11-20	DINF(2)	FFSPEC F10.0	X direction cosine component of local velocity vector ( $V_x/V$ )
21-30	DINF(3)	FFSPEC F10.0	Y direction cosine component of local velocity vector ( $V_y/V$ )
31-40	DINF(4)	FFSPEC F10.0	Z direction cosine component of local velocity vector ( $V_z/V$ )
41-50	DINF(5)	FFSPEC F10.0	Flow generating turning angle from freestream direction required to give desired local flowfield.
51-60	DINF(6)	FFSPEC F10.0	Not Used

**Hand Load Surface Data:** These cards are used to hand load surface pressures directly on the flowfield storage unit (10). These cards are input only if IDTYP(1) = 0. The Hand Load Surface Data cards have been modified to ease the entry of experimental data. Pressure data can now be loaded in any order without concern about subregions. In addition, unnecessary inputs have been eliminated. The new cards required for the Hand Load Surface Data Option are shown below.

**Hand Load Surface Data Control Card (I3)**

Column	Variable	Routine Format	Explanation
1-3	NPTS	FFSURF I3	Number of surface data points to be loaded

**Hand Load Surface Data Card (4F10.0)**

Column	Variable	Routine Format	Explanation
1-10	SDATA(1)	FFSURF F10.0	X coordinate of surface data point
11-20	SDATA(2)	FFSURF F10.0	Y coordinate of surface data point
21-30	SDATA(3)	FFSURF F10.0	Z coordinate of surface data point
31-40	SDATA(4)	FFSURF F10.0	Pressure Coefficient at surface data point NOTE: This is the pressure coefficient and not pressure ratio.

Note: Need NPTS number of these cards

### A4.3 Shielding Option Input Data

The Shielding Program may be used to account for the situation where one part of a vehicle shape is shielded from the freestream flow by another part of the vehicle. The geometry data for use by the Shielding program must be stored and available on the Quadrilateral Element Storage Unit (4). The Geometry Options must be used to accomplish this.

Before the Shielding Program is used on a given vehicle, the user should view the configuration at each of the  $\alpha$ - $\beta$  conditions to be analyzed for shielding. This will provide the user with information as to what part of the vehicle is being shielded by what other parts. From these views, the user should make a list of each panel of the vehicle that will experience some shielding. For each shielded panel, a list should also be made of what other vehicle panels cause the shielding. These lists will be input on the shielding input data cards to reduce the amount of time that will be required to perform the shielding searches.

The Shielding Program will perform its shielding search and will generate and store a special set of quadrilateral elements. These special quadrilateral elements will have negative surface areas and taken all together will represent those parts of the vehicle that are shielded by some other upstream part. The negative area shielded elements are stored on Unit 3. One set of negative area elements will be stored for each  $\alpha$ - $\beta$  specified on the input to the AERO executive routine. When the FORCE program calculates the pressures on the vehicle, it will proceed in a normal manner until all of the regular vehicle elements are accounted for. It will then turn to the negative area shielded elements and calculate the pressures also in a perfectly usual manner, except that the element areas used will be negative. In this manner, the shielded parts of the vehicle will be effectively removed from the analysis and will make no contribution to the final vehicle forces.

#### Shielding Title Control Card (I2, I1, I2, 15A4)

Column	Variable	Routine Format	Explanation
1-2	NPANL	SHIELD I2	The total number of PANELS to be considered and analyzed for shielding.
3	IPRINT	SHIELD I1	Print flag. = 0 Do not print negative area shielded elements. = 1 Print characteristics of the negative area quadrilateral elements.
4-5	INAB	SHIELD I2	Angle of attack analysis control flag. (not used in present program)
6-65	TITLE	SHIELD 15A4	Title for printout on shielding print pages.

Shielding Panel Control Card (I2, 20I2)

Column	Variable	Routine Format	Explanation
1-2	IPAN	SHIELD I2	The sequence number of the Panel to be considered for shielding.
3-4 5-6 7-8 etc.	ISHE (1) ISHE (2) ISHE (3)	SHIELD 20I2	The sequence number of the Panel that is to be considered as a possible shielding panel. Up to a total of 20 such Panel numbers may be specified on each of these Shielding Panel Control Cards.

NOTE: The number of these cards to be read must be equal to NPANL as specified on the Shielding Title Control Card.

#### A4.4 Inviscid Pressure Option Input Data

The Pressure Calculation Program is reached by way of sub-option calls from the Aero Executive routine. The sequence of calls to the FORCE routine and the saving and summation of force data is accomplished in routine PRES. The geometry data for use by the PRES and FORCE routines must be sorted and available on the quadrilateral Element Storage unit (4). The Geometry Options must be used to accomplish this storage. An input to routine PRES specifies how the vehicle Panels are to be grouped to form vehicle components for the force analysis. These selections are made by using the Panel sequence numbers assigned by the Geometry program (the panels are numbered in the order in which they are placed on the Quadrilateral Element Storage Unit (4)).

##### Title Card and Basic Flags (I2, I1, 3X, 15A4)

Column	Variable	Routine Format	Explanation
1-2	NCOMP	PRES I2	Total number of vehicle components to be analyzed. Each component may consist of one or more vehicle panels. The grouping of panels to form components is controlled by the Component Organization Card below. (20 maximum)
3	IFSAVE	PRES I1	Force data save flag. = 0 Set up a new Unit 9 Force Data save file. Save component force data. = 1 Save force data. Use existing Unit 9 Force Data file and just add a new set to it. = 2 Do not save force data.
7-66	TITLE	PRES 15A4	Title to be printed out at the top of the force data output.

##### Component Organization Card (I3, 3I1, I2, F10.4)

Column	Variable	Routine Format	Explanation
1-3	ICOMP	PRES I3	Component number for analysis.
4	IPM	PRES I1	Pressure Method card flag. = 0 Pressure Method cards will be read for each $\alpha$ - $\beta$ . = 1 Read only one Pressure Method card and assume that it will apply for all the $\alpha$ - $\beta$ s for this component. = 2 Use the same pressure method data set as was used for the previous vehicle component. No Pressure Method cards will be input. = 3 Input $C_p$ table will be used. Input Pressure Option Cards will be input. Do not input Pressure Method Cards. This option may be used to input wind tunnel pressure data in order to obtain resultant vehicle forces. = 4 The non-uniform $C_p$ table that has been previously generated by the Second-Order Shock-Expansion method (of the Flow Field Option) will be used over the surface of the vehicle component. Do not input Pressure Method Cards. Input Pressure Option Cards will be expected. This is the way provided in the program in which the Second-Order Shock-Expansion method can be used as a pressure calculation method.

5	INT	PRES I1	Interference method card flag. = 0 No interference cards will be read. Interference effects will not be accounted for in the force calculations. = 1 Interference method cards will be read for each $\alpha$ - $\beta$ for this component. = 2 Read only one interference method card and assume that it will apply for all the $\alpha$ - $\beta$ s for this component. = 3 Use the same interference method data set as was used for the previous vehicle component. No interference method cards will be input.
6	ISHEF	PRES I1	Shielding Elements flag. = 0 Shielding elements have not been generated for this component and shielding effects will therefore not be accounted for. = 1 Shielding elements have been generated and are stored on Unit 3. Shielding effects should be accounted for on this vehicle component.
7-8	IGTYPE	PRES I2	= 0 Fixed surface = 3 Pitch control surface, symmetric control deflection = -3 Yaw/Roll control surface, asymmetric control deflection
9-18	CDEL	PRES F10.0	Deflection angle for this component (deg).

Note: The number of sets of Component Organization, Pressure Method Cards, and Interference Method Cards must be = NCOMP.

#### Hinge Line Card (6F10.0)

This card is read only if IGTYPE = 3 or -3 on the Component Organization Card.

Column	Variable	Routine Format	Explanation
1-10	X1	READL1 F10.0	X-Coordinate of hinge line at tip. (Input using scaled coordinates)
11-20	Y1	READL1 F10.0	Y-Coordinate of hinge line at tip. (Input using scaled coordinates)
21-30	Z1	READL1 F10.0	Z-Coordinate of hinge line at tip. (Input using scaled coordinates)
31-40	X2	READL1 F10.0	X-Coordinate of hinge line at root. (Input using scaled coordinates)
41-50	Y2	READL1 F10.0	Y-Coordinate of hinge line at root. (Input using scaled coordinates)
51-60	Z2	READL1 F10.0	Z-Coordinate of hinge line at root. (Input using scaled coordinates)

Note: If surface is on right side (i.e., negative y values) then input root first.

Pressure Method Cards (2I2, 3I1, 3X, 6F10.0)

The Pressure Method Cards are used to specify what pressure calculation methods are to be used for each vehicle component. The necessary constants for each pressure method are also input on these cards. The number of Pressure Method Cards to be input is controlled by the parameter IPM on the Component Organization Card. If IPM = 0 then the number of Pressure Method Cards must be equal to the parameter NAB as input on the Flight Condition Card to the AERO executive routine

Column	Variable	Routine Format	Explanation
1-2	IMPACT	PRES I2	Impact force-calculation method flag. The following methods are available for calculation of pressures on surface elements in impact flow (right-justified integer). = 1 Modified Newtonian (K is input in CC 11-20). = 2 Modified Newtonian + Prandtl-Meyer (CC 31-40 must contain the proper value for $\eta_c$ ). = 3 Tangent-wedge (using oblique-shock). = 4 Tangent-wedge empirical. = 5 Tangent-cone. = 6 Inclined-cone method. = 7 Van Dyke Unified Method (small disturbance theory). = 8 Not Used = 9 Shock-expansion Method using strip theory. The parameter IORN on the Panel Identification Card in the Geometry Option identifies which edge of the panel is the leading edge. IORN may be = 0 or 1 only. =10 Free-molecular flow. Input $f_n$ in CC 11-20 $f_t$ in CC 41-50, and $T_B/T_\infty$ in CC 31-40. =11 Input constant pressure coefficient (use CC 11-20 for the pressure coefficient). A constant pressure coefficient will be applied over all elements. =12 Not Used =13 Not Used =14 Dahlem-Buck empirical. =15 Not Used
3-4	ISHAD	PRES I2	Shadow force-calculation method flag. The following methods are available for calculation of pressures on surface elements in shadow flow (right-justified integer). = 1 Newtonian (i.e., $C_p = 0.0$ ). = 2 Modified Newtonian + Prandtl-Meyer (CC 31-40 must contain the proper value for $\eta_c$ ). = 3 Prandtl-Meyer expansion from free-stream = 4 Inclined cone method. = 5 Van Dyke Unified Method (small disturbance). = 6 High Mach number base pressure ( $C_p = -1/M^2$ ). = 7 Shock-expansion (strip theory). See IMPACT = 9 discussion. = 8 Input pressure coefficient (use CC 31-40 for the input pressure coefficient). = 9 Free-molecular flow. See IMPACT = 10 for other input requirements. =10 Dahlem - Buck Mirror =11 ACM Empirical =12 Half Prandtl-Meyer



# Appendix A, Input Data Instructions

5	IPRINT	PRES I2	Print flag. This flag controls the printing of the detailed force characteristics of each vehicle element. = 0 Do not print detailed element force data. = 1 Print detailed force contributions for each element (a large amount of output will be produced and machine time will increase). = 2 Print detailed local property calculation and iteration results.
6	IPIN	PRES I1	Not used in current program. Use IPM flag on Component Organization Card.
7	ISAVE	PRES I1	Save surface property data flag. = 0 Do not save the surface property data (local surface Mach number, $P/P_\infty$ , $T/T_\infty$ , etc.). = 1 Save surface property data on unit 4. <b>This option must be used to store data that is required by the skin friction options.</b> These data may also be used in the Surface Data Transfer Option.
11-20	PDATA (1)	PRES F10.0	Pressure method input parameters. The input parameter in this field will vary depending upon the pressure method option selected. For IMPACT = 1, 2, or 3 input the modified Newtonian correction factor, K (CPSTAG). For IMPACT = 10 input the free-molecular flow parameter, $f_n$ . For IMPACT = 11 input a constant pressure coefficient, $C_p$ .
21-30	PDATA (2)	PRES F10.0	QQINF. Dynamic pressure (q) at the surface divided by the freestream q.  $C_{p\infty} = C_p (q/q_\infty)$  <b>Must be input as 1.0 if no change from free-stream is to be made.</b> This parameter is useful in removing the effect of a vehicle component or in changing the local q for a whole component because of a constant $q/q_\infty$ effect of an interference component.
31-40	PDATA (3)	PRES F10.0	This field is used for several different input parameters depending upon the values of the impact and shadow pressure calculation method flags.  = Prandtl-Meyer expansion correction factor $\eta_c$ (ETAC) in the following equation.  $C_p = \frac{P\eta_c - P_\infty}{q_\infty}$  This is used when IMPACT = 2 or ISHAD = 2, but is usually input as 1.0. = Input pressure coefficient in shadow regions when ISHAD = 8. = $T_B/T_\infty$ for IMPACT = 10. $T_B/T_\infty$ is the ratio of body temperature to freestream temperature.

# Appendix A, Input Data Instructions

41-50	PDATA (4)	PRES F10.0	<p>ENPM. Surface slope modification factor. If input as <math>\neq 0.0</math> the surface slope (<math>\theta</math>, angle between outward surface normal and velocity vector) will be divided by this number. The impact angle (<math>\delta</math>) is calculated as follows:</p> $\delta = \frac{\pi}{2} - \frac{\theta_{input}}{ENPM}$ <p>If ENPM is input as -0.0 or 1.0, then the body slope is not changed.</p> <p>This location has an alternate use when IMPACT = 10.  <math>= f_t</math> (tangential momentum accommodation coefficient, -0.0 for Newtonian flow and 1.0 for completely diffuse reflection.)</p>
51-60	PDATA (5)	PRES F10.0	<p>IMPACI. Impact method for Shock-expansion calculations. This flag controls the method to be used in the calculation of the pressure and local properties on the first element of each streamwise strip for subsequent shock-expansion calculations. The available methods are listed below. This field is used only when IMPACT = 9.</p> <p>= 3.0 Tangent-wedge (oblique shock).          = 5.0 Tangent-cone.          = 13.0 Delta-wing empirical.</p>
61-70	PDATA (6)	PRES F10.0	<p>ISHADI. Shadow method for Shock-Expansion calculations. This flag controls the method to be used in the calculation of the pressure and local properties on the first element of each streamwise strip for subsequent shock-expansion calculations (if the first element is in a shadow region). This field is used only when ISHAD = 7. The only acceptable method at the present time is</p> <p>= 3.0 Prandtl-Meyer expansion from freestream.</p>

**Flowfield Interference:** The Flowfield Interference Method Cards are used to specify the type of interference computations that are to be used for each vehicle component. The number of Interference Method Cards to be input is controlled by the parameter INT on the Component Organization Card. The Interference Method Cards allows the user to use flowfield data previously loaded on the flowfield storage unit 10, or to input a uniform flowfield using the Uniform Flowfield Card. The Flowfield Interference Card has been modified to account for the new interpolation method implemented in S/HABP Mark V. The number of inputs have been substantially reduced and are more understandable. The only new input is the radius of influence which is used to control the number of points used in the interpolation. The interpolation method uses a simple distance weighted average to determine the flow properties at any given point within the flowfield.

Flowfield Interference Method Control Data Card (9I1,1x,F10.0)

Column	Variable	Routine Format	Explanation
1	INF(1)	PRES I1	Interference Data Source Flag = 0 Uniform flowfield will be hand input using the Uniform Flowfield Data Card = 1 Uniform flowfield data will be obtained from unit 10 = 2 Non-Uniform flowfield data will be obtained from unit 10
2	INF(2)	PRES I1	Flowfield data set number where data on unit 10 is located.
3	INF(3)	PRES I1	Number of flow regions to be used to obtain flowfield data.
4	INF(4)	PRES I2	First flowfield region number
5	INF(5)	PRES I1	Second flowfield region number
6	INF(6)	PRES I2	Third flowfield region number
7	INF(7)	PRES I2	Fourth flowfield region number
8	INF(8)	PRES I2	Fifth flowfield region number
9	INF(9)	PRES I2	Detail Print Flag = 0 Do Not Print Interpolated Data = 1 Print Interpolated Flowfield Data
11-20	FFRAD	PRES F10.0	Radius of influence (All points within the specified radius will be used to interpolate for the local flowfield properties)

Uniform Flowfield Data Card (6F10.0)

Column	Variable	Routine Format	Explanation
1-10	DINF(1)	FFINPT F10.0	Local Mach Number ( $M_{\text{local}}$ )
11-20	DINF(2)	FFINPT F10.0	X direction cosine component of local velocity vector ( $V_x/V$ )
21-30	DINF(3)	FFINPT F10.0	Y direction cosine component of local velocity vector ( $V_y/V$ )
31-40	DINF(4)	FFINPT F10.0	Z direction cosine component of local velocity vector ( $V_z/V$ )
41-50	DINF(5)	FFINPT F10.0	Local Pressure Ratio ( $P/P_\infty$ ) <sub>local</sub>
51-60	DINF(6)	FFINPT F10.0	Local Temperature Ratio ( $T/T_\infty$ ) <sub>local</sub>

Input Pressure Option: The Input Pressure Option Cards are used when the vehicle component forces are to be calculated using pressure data previously stored on the flowfield data unit 10. These cards are only input if IPM = 3 or 4. This option may be used in several ways. For example, the forces on a particular component may be calculated using experimental data results which have been previously stored on unit 10. More directly, forces may be determined using the data generated by the Shock Expansion Flowfield Option and stored on unit 10. The Input Pressure Option Cards have been modified to account for the new surface spline method implemented in S/HABP Mark V. The number of inputs have been substantially reduced and are more understandable. The only new input is the radius of influence which is used to control the number of points used in the surface spline. It is recommended that the radius of influence be set such that a minimum of eight points are available for interpolation.

Input Pressure Option Control Card (5I1,5x,F10.0)

Column	Variable	Routine Format	Explanation
1	CPSET	PRES I1	Flowfield data set number where data is located on unit 10.
2	CPR	PRES I1	Region number where data is located on unit 10.
3	IPRINT	PRES I1	Print Flag. This flag controls the printing of the detailed force characteristics of each vehicle element. = 0 Do not print detailed element force data. = 1 Print detailed force contributions for element. (a large amount of output will be generated) = 2 Print detailed local property calculation and iteration results. (Note: This is the same flag as on Pressure Method Card)
4	ISAVE	PRES I1	Save Surface Property Data Flag = 0 Do not save the surface property data (local Mach number, $P/P_\infty$ , $T/T_\infty$ , etc.) = 1 Save surface property data on unit 4. (Note: This is the same flag as on Pressure Method Card)
5	INTPRT	PRES I1	Print interpolated pressure data = 0 Do not print interpolated data = 1 Print interpolated pressure for each element
11-20	CPRAD	PRES F10.0	Radius of influence (All points within the specified radius will be used to determine the pressure coefficient at each element centroid)

### A4.5 Viscous Analysis Option

The Viscous Program Option is reached by way of sub-option calls from the AERO executive routine. Routine VISCUS is the control routine for the viscous calculations and is similar to the PRES routine used in the Inviscid Option of the program. The Viscous Option makes use of surface property data (local pressure, temperature, and Mach number) that has been previously calculated by the Inviscid FORCE routine for the geometry being studied.

#### Skin Friction Basic Flag and Title Card (I2, I1, 3X, 15A4)

Column	Variable	Routine Format	Explanation
1-2	NCOMP	VISCUS I2	Total number of vehicle components to be analyzed. Each component may consist of one or more vehicle panels. The grouping of panels to form components is controlled by the Geometry Data Source Card.
3	IFSAVE	VISCUS I1	Force data save flag. = 0 Set up a new force data save file (unit 9). Save skin friction force data for future summation. = 1 Save skin friction force data for future summation on unit 9. Use old unit 9 file and just add the new force data on to the file. = 2 Do not place the force data on the force data file unit.
7-66	TITLE	VISCUS 15A4	Title to be printed out on the skin friction output pages.

#### Geometry Data Source Card (I3, I1, I3)

Column	Variable	Routine Format	Explanation
1-3	ICOMP	VISCUS I3	Component number to be analyzed.
4	ISK	VISCUS I1	Skin Friction Method Card Flag. = 0 Skin Friction Method cards will be read for each $\alpha$ - $\beta$ . = 1 Read only one Skin Friction Method Card and assume that it will apply for all of the $\alpha$ - $\beta$ s for this component. = 2 Use the same Skin Friction Method data set as was used for the previous vehicle component (no Skin Friction Method Cards will be read).
5-7	NS	VISCUS I3	Number of skin friction elements to be analyzed. This number must be equal to the number of elements on the Quadrilateral Element save unit 4 for this vehicle component and must not be greater than 100. The number of Skin Friction Element Data Cards must be = NS. This input is used for the Mark III skin friction option only.

Skin Friction Method Cards (3I1)

These cards control the method to be used in calculating the skin friction coefficients used in the skin friction computations. The number of these cards is controlled by the ISK flag on the Component Organization Card.

Column	Variable	Routine Format	Explanation
1	ISFMTH	VISCUS I1	Skin friction method flag. = 0 Level 2 Viscous. = 1 Level 1 Viscous (Calculate skin friction coefficients using Mark III methods).
2	IPRINT	VISCUS I1	Print flag when Mark III skin friction methods have been used (ISFMTH = 1). = 0 Do not print. = 1 Print detailed skin friction computation intermediate results.
3	ISAVE	VISCUS I1	Skin friction coefficient data save flag. = 0 Do not save. = 1 Write skin friction coefficient for each element on unit 4 along with the previously saved surface data. This option only used if ISFMTH = 0.

### A4.5.1 Level 1 Viscous Analysis

#### Level 1 Skin Friction Element Data Cards (I2, 8I1, 2F9.0, 3F6.0, 2F6.0, F4.0, 8X, I2)

One Skin Friction Element Data Card must be loaded for each element stored on the Quadrilateral Element Storage Unit (4) for each vehicle component. The format of these cards is exactly the same as the Type 11 cards used on the Mark III program (Mode 1 skin friction method). However, some of the parameters on the old Type 11 card are not actually used by this new version of the program.

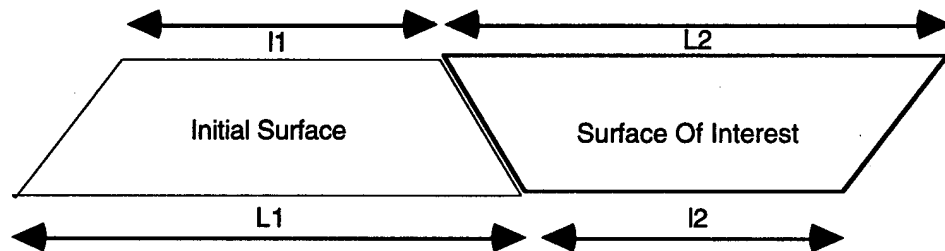
Column	Variable	Routine Format	Explanation
1-2	IS (I,1)	SKINFR I2	Skin friction element number.
3	IS (I,2)	M3SF I1	Viscous-Inviscid interaction effect flag. = 0 Use tangent-wedge in interaction correction. = 1 Use tangent-cone in interaction correction
4	IS(I,3)	SKINFR I1	Calculate induced pressures due to boundary layer displacement effects. Skin friction is not calculated. = 0 No = 1 Yes
5	IS (I,4)	SKINFR I1	Skin-friction summation flag. = 0 Use turbulent skin friction data in calculating forces. (Note: The program will make a switch to laminar summation at very low Reynolds number, where turbulent results are not meaningful). = 1 Use laminar skin friction data in calculating forces.
6	IS (I,5)	SKINFR I1	(Not used in this program).
7	IS (I,6)	SKINFR I1	Wall-temperature and skin-friction method Flag. The program always calculates both laminar and turbulent skin-friction results. The result to be added to the pressure calculations is indicated by the flag in CC 5. In the discussions below, the methods to be used for laminar and turbulent flow are separated by a slash (i.e., Laminar/Turbulent). = 0 Calculate wall temperature and skin friction using Reference Temperature/Spalding-Chi methods = 1 Use adiabatic wall temperature and Reference Temperature/Spalding-Chi methods. = 2 Use input wall temperature and Reference Temperature/Spalding-Chi methods. $T_w$ input in CC 47-52 and 53-58. = 3 Calculate wall temperature and skin friction using Reference Enthalpy/Spalding-Chi (with enthalpy ratios) methods. = 4 Use adiabatic wall temperature and Reference Enthalpy/Spalding-Chi (with enthalpy ratios) methods.

# Appendix A, Input Data Instructions

			<ul style="list-style-type: none"> <li>= 5 Use input wall temperature and Reference Enthalpy/Spalding-Chi (with enthalpy ratios) methods. <math>T_w</math> input in CC 47-52 and 53-58.</li> <li>= 6 Calculate wall temperature and skin friction using Reference Temperature/Reference Temperature methods.</li> <li>= 7 Use input wall temperature and Reference Temperature/Reference Temperature methods. <math>T_w</math> input in CC 47-52 and 53-58.</li> <li>= 8 Calculate wall temperature and skin friction using Reference Enthalpy/Reference Enthalpy methods.</li> <li>= 9 Use input wall temperature and Reference Enthalpy/Reference Enthalpy methods. <math>T_w</math> input in CC 47-52 and 53-58.</li> </ul>
8	IS (I,7)	SKINFR I1	Flag to control printing of skin-friction data for each skin-friction surface element. = 0 Do not print. = 1 Print skin-friction data. This is recommended option for most applications.
9	IS (I, 8)	SKINFR I1	Not Used
10	IS (I,9)	SKINFR I1	Iteration and local skin friction print flag. = 0 Do not print. = 1 Print iteration results for wall temperature and the final local skin-friction data. = 2 Print the final local skin-friction data, but not the iteration results. This is the recommended option for most applications.
11-19	SURF (I,1)	SKINFR F9.0	Skin friction element surface wetted area in same units as $S_{ref}$ . If input as 0.0 then the program will use the surface area as calculated from the input geometry unit for each element. The input wetted area must correspond to the input skin-friction geometry (i.e., if the Symmetry flag is 0, left side of the vehicle input, then the input wetted area should be only for the left side).



The four input quantities in CC 20 through 46 furnish to the program the planform shape of the skin-friction surface being analyzed ("Surface-of-Interest"), and the shape of the initial-surface (to account for the fact that the flow has traversed some other part of the shape before reaching the surface of interest). This information is not obtained from the input skin-friction geometry data input on the Type 3 cards. The input skin-friction geometry data are used only to establish the position and orientation of the centroid and the area of each skin-friction surface. The diagram below illustrates the input parameters required on the Skin Friction Element Data cards.



20-28	SURF (I,2)	SKINFR F9.0	The longest length of the surface-of-interest ( $L_2$ in the diagram above). If input as 0.0, the program will use the surface area as calculated from the input geometry unit for each element. Feet.
29-34	SURF (I,3)	SKINFR F6.0	The longest length of the initial-surface ( $L_1$ in the diagram above). Feet.
35-40	SURF (I,4)	SKINFR F6.0	The taper ratio of the initial-surface ( $I_1/L_1$ ). This taper ratio is defined as the ratio of the shortest chord length to the longest chord length. If both the initial-surface longest-length and the longest length of the surface-of-interest are on the same edge of the shape, then the taper ratio of the initial-surface is input as a positive number. If these lengths are on opposite sides of the shape such as in the diagram above, then the initial surface taper ratio is input as a negative number. With these ground rules, the absolute value of the taper ratio will never be greater than 1.0.
41-46	SURF (I,5)	SKINFR F6.0	The taper ratio of the surface-of-interest ( $I_2/L_2$ ). This taper ratio is defined as the ratio of the shortest chord length. This taper ratio is always positive and never greater than 1.0. If input as 0.0, the program will use the surface area as calculated from the input geometry unit for each element.
47-52	SURF (I,6)	SKINFR F6.0	Input wall temperature for laminar calculations, °R. This input is used when CC 7 = 2, 5, 7, or 9.
53-58	SURF (I,7)	SKINFR F6.0	Input wall temperature for turbulent calculations, °R. This input is used when CC 7 = 2, 5, 7, or 9.
59-62	SURF (I,8)	SKINFR F4.0	(Not used in present program).
71-72	TYPE	SKINFR I2	Card Type number 11.

#### A4.5.2 Level 2 Viscous Analysis

Level 2 Viscous analysis applies engineering methods along streamlines predicted by the VECC GUI for enhanced accuracy. This section provides detail on input required for the level 2 viscous analysis.

Subroutine SETUP sets the constants and user inputs required for the level 2 viscous analysis. First it uses data statements to initialize the constants and the level 2 viscous input defaults. Next it reads the level 2 viscous inputs from the *.mk5* file, and stores them for later use during the streamline analysis. Finally, it updates the flow medium parameters based upon the input read from the *.mk5* file.

The information within the level 2 viscous input is organized per component both in the GUI and in the *.mk5* input file.

The third card contains the inviscid pressure method information entered in the top right corner of the level 2 viscous window. The GUI and the card itself are exactly the same as the inviscid *Pressure Method* card. The first variable specifies the windward pressure method, the second, the leeward method. The next three variables, flags for saving and printing data, are not used in the viscous method. The last six variables are constants for each pressure method.

The remaining three cards contains the user flags and constants for the boundary layer analysis of the components. The input can be divided into six general groups:

1. Boundary Condition
2. Gas Equation
3. Boundary Layer State
4. Boundary Layer Method
5. Separation
6. Output

For the boundary condition, the user may specify either cold wall (constant wall temperature) or radiation equilibrium temperature wall condition. In addition, the wall temperature must be specified if the cold wall condition is chosen. On the other hand, the wall emissivity and the radiation sink temperature are required for the radiation equilibrium wall condition.

For the gas equation, the user may specify ideal gas, effective gamma, or real gas.

The boundary layer state include laminar, transitional and turbulent flow. Transitional flow contains six transition criteria for specifying the onset and fully turbulent conditions. Except for two of the criteria, the user must specify the onset and fully turbulent values.

The boundary layer model specifies the correlation method for analyzing the component's skin friction and heat transfer. Both flat plate and leading edge correlations are available. For the flat plate option, the users selects the desired laminar and turbulent correlations and set the cone correction flag. For the leading edge option, the user selects the leading edge type and specifies the nose radius. The leading edge types include sphere, cylinder and swept cylinder. For the swept cylinder option, the user must also specify the sweep and elevation angle. In addition, two correlations are available for the swept cylinder leading edge.

The separation input includes the separation criteria flag and the corresponding onset value. One of the criteria specifies no separation , and requires no separation onset value.

Output for the level 2 analysis is controlled by three print flags. The user may output results along the streamlines, results interpolated onto the element centroids, and graphical output. The streamline and element results are tabulated in the output file. The graphical output consist of a VISCOUS.TP3.# geometry file and an VISCOUS.ERF.# results file. These file are used with VUAERO for graphical viewing.

All the flags are grouped into the first card, and the constants are grouped into the last two cards. The input for the boundary layer analysis these cards are explained in the following tables:

#### Boundary Layer Flag Card (15I2)

Column	Variable	Routine Format	Explanation
1-2	mgas	SETUP I2	Gas Equation Flag = 0 Ideal gas equation used to calculate thermodynamic properties of the flow medium. = 1 Ideal gas equation with effective gamma assumption. = 2 Real gas (equilibrium air) routines.
3-4	jlam	SETUP I2	Boundary Layer State Flag = 0 Transitional Flow = 1 Laminar Flow = 2 Turbulent low
5-6	jtrans	SETUP I2	Transition Criteria Flag = 1 Running Length = 2 Reynolds Number = 3 Momentum Thickness Reynolds Number = 4 Momentum Thickness Reynolds Number / Edge Mach Number = 5 BLT-1A = 6 BLT-2D
7-8	jheat	SETUP I2	Boundary Layer Model Flag = 1 Leading Edge Method = 2 Flat Plate Method
9-10	jplate	SETUP I2	Cone Correction Flag = 1 Flat Plate (2D Flow) = 2 Cone (Axisymmetric Flow)
11-12	iplatl	SETUP I2	Laminar Flat Plate Method Flag = 1 Eckerts Reference Enthalpy Method = 2 Boeing Rho-Mu Method

## Appendix A, Input Data Instructions

13-14	iplatt	SETUP I2	Turbulent Flat Plate Method Flag = 1 Schultz-Grunow Method = 2 Boeing Rho-Mu Method = 3 Spalding-Chi Method = 4 White Method
15-16	jledgs	SETUP I2	Leading Edge Type Flag = 1 Sphere = 2 Cylinder = 3 Swept Cylinder
17-18	jledgl	SETUP I2	Laminar Leading Edge Method Flag = 1 Not Swept Cylinder = 2 Cato-Johnson Swept Cylinder = 3 Beckwith-Gallagher Swept Cylinder
19-20	jledgt	SETUP I2	Turbulent Leading Edge Method Flag = 1 Not Swept Cylinder = 2 Cato-Johnson Swept Cylinder = 3 Beckwith-Gallagher Swept Cylinder
21-22	jwllbc	SETUP I2	Wall Boundary Condition Flag = 1 Cold Wall = 2 Radiation Equilibrium Temperature Wall
23-24	jelprt	SETUP I2	Interpolated Elemental Output Print Flag = 0 Do not print output = 1 Print output
25-26	jgmprt	SETUP I2	Viscous Geometry/ERF Print Flag = 0 Do not print .TP3 and >ERF files = 1 Print .TP3 and .ERF files
27-28	jseprt	SETUP I2	Separation Onset Criteria Flag = 1 X-Station = 2 Wetted Distance = 3 Phi angle from bottom centerline = 4 Reynolds Number
29-30	jstprt	SETUP I2	Streamline Result Output Print Flag. If set results of <b>all</b> streamlines will be printed. = 0 Do not print streamline results = 1 Print streamline results

### Boundary Layer Constant Card I (5 (1x, E12.6))

Column	Variable	Routine Format	Explanation
2-13	onst	SETUP E12.6	Onset of transition value associated with the selected transition criteria (jtrans = 1-4). Not required by the NASP criteria (jtrans = 5,6). (in geometry units)
15-26	trb	SETUP E12.6	Fully turbulent value associated with the selected transition criteria (jtrans = 1-4). Not required by the NASP criteria (jtrans = 5,6). (in geometry units)
28-39	xrnose	SETUP E12.6	Leading edge nose radius (in geometry units)
41-52	fins	SETUP E12.6	Swept cylinder leading edge sweep angle (-90 to + 90 degrees measured from $y=0^\circ$ )
54-65	finp	SETUP E12.6	Swept cylinder leading edge elevation angle (0 to 180 degrees measured from $-z = 0^\circ$ )

Boundary Layer Constant Card II (5 (1x, E12.6))

Column	Variable	Routine Format	Explanation
2-13	xtwall	SETUP E12.6	User specified wall temperature (R)
15-26	xemise	SETUP E12.6	User specified wall emissivity
28-39	xrdinf	SETUP E12.6	User specified radius of influence. Only results from streamline points within this radius are used for the interpolation to the element centroid (in geometry units)
41-52	xtsnke	SETUP E12.6	User specified radiation sink temperature (R)
54-65	xonspt	SETUP E12.6	User specified value for onset of separation (in geometry units)

### A4.6 Special Routines Option Input Data

The Special Routines section of the program is reached by way of suboption calls from the AERO Executive routine. The Special Routines routine currently has two options; the Summation routine, and the Pressure Output routine. The basic purpose of the Summation routine is to add together the aerodynamic force coefficients of the separate vehicle components as calculated in the Force and Skin Friction programs. The components to be added are selected by save-set number (sequence number in which they were placed on to the Force Data unit 9). The Pressure Output routine allows the user to write a file containing pressure coefficient data for each element in a format compatible with the VUAERO program.

#### Special Option Selection Card (20I1)

Column	Variable	Routine Format	Explanation
1	IPG (1)	SPEC	Special Routines sub-options to be used in the order given.  = 1 Summation Program will be called. This program may be used to add, print, and punch force data saved on the Force Data Save unit (9). = 2 NOT USED = 3 Pressure Output for VUAERO
2	IPG (2)	20I1	
3	IPG (3)		
.	.		
.	.		
etc.	etc.		
20	IPG (20)		

#### A4.6.1 Summation Routine Option

This routine may be used to selectively sum and print data that has been stored on the Force Data Save unit (9) by the Force and Skin Friction routines. The results of the summations may also be stored on the Force Data Save unit. This is the only option provided in the program for adding together the force contributions of different vehicle components.

Summation Control Card (5I1, I4, 20I2)

Column	Variable	Routine Format	Explanation
1	LAST	SUM I1	Last control card flag. = 0 This is not the last control card. Another Control card will be expected next. = 1 This is the last control card. Return to the Aero routine after this card set of sum/print/punch instructions is completed.
2	ISUM	SUM I1	Summation flag. = 0 Do not sum up component forces. = 1 Sum up all the vehicle components as specified by the ICOMP parameter.
3	ISAVE	SUM I1	Data save flag. = 0 Save summation data on the Force Data Save unit (9). = 1 Do not save summation data.
4	IPRINT	SUM I1	Print control flag. = 0 Print out summation data only. Component data will not be printed. = 1 Print component and summation data. Print component data even if summation was not performed.
5	IPUNCH	SUM I1	Punch control flag. = 0 Do not print any data. Always zero. = 1 Print summation data only. = 2 Print both component and summation data.  The data will be printed to unit 7 in the following order on a single line. ALPHA, CN, CA, CM, CY, CLN, CLL, BETA, IRUN The card format is as follows: FORMAT (7F9.4, F7.2, 1X, I4) One identification card is printed for each set.
6-9	IRUN	SUM I4	Run number to be printed.
10-11 12-13 14-15 . . . etc. 48-49	ICOMP (1) ICOMP (2) ICOMP (3) . . . etc. ICOMP (20)	SUM 20I2	Selected component sequence numbers to be summed or printed. These numbers must correspond to the order of the data placed on the Force Data Save unit (9) in the Force and Skin Friction routines. For example, if you wish to sum up components number 5, 7, and 11, then input ICOMP (1) = 5, ICOMP (2)=7, and ICOMP (3)=11.

If LAST = 0 then another Summation Control Card will be expected next.

If LAST = 1 then the return to AERO will be called next.

### A4.6.2 Surface Pressure Output

#### Pressure Data Transfer Card (I1, I3)

This option gives the user the capability to transfer pressure data calculated using the inviscid pressure option and saved on unit 4 (ISAVE = 1 on Pressure Method Card) to a plot file for use by the VUAERO postprocessor. One card is input for each component to be written to the pressure file (remember to set LAST=1 on the final component to be written). If a component is analyzed more than once in a case and ISAVE=1 on the *Pressure Method Card*, the pressure data for the last analysis will be written. If the user desires to display the difference in pressures due to different pressure methods on the same component or components, each analysis should be run in a different case. Two files are written when this option is used. The first file named PRES.GEO.# contains the panels for which the user has requested pressure data. The second file, PRES.DAT.#, contains the pressure data for each  $\alpha$ - $\beta$  analyzed. The final extension # is a number which is incremented for each case in which pressure data is requested.

Column	Variable	Routine Format	Explanation
1	LAST	PRESOT I1	Last control card for transfer. = 0 This is not the last control card, another card will be read. = 1 This is the last control card.
2-4	ICOMP	PRESOT I3	Component number



## A5. AUXILIARY PROGRAMS

This program option is reached by way of System Option calls from the Main Executive Routine *System Control Card* (IPG = 3). The Auxiliary Programs feature is provided as a means of attaching additional programs or features to the Mark V program framework as might be required by the user and is not supported in the graphical user interface (GUI). If these routines are to be accessed, the user must manually (i.e. in a text editor) enter IPG=3 in the *System Control Card*. The cards defined in this section must then be placed in the order determined by the placement of the IPG=3 flag in the *System Control Card*. For example if printing the stability and control output file, IPG would be set to 3 after all of the calls to the aerodynamic program. Therefore, the *Auxiliary Programs Control Card* would be placed after the last Aerodynamic Program input card, and before the component names. The file will not be compatible with the GUI when using the auxiliary programs. The "Stability and Control Output" option listed below makes Mark V compatible with the Hypervelocity Stability and Control post-processor, POSTSC (References 1 and 2) which was developed under a separate contract. The "General Cutting Plane" option is no longer necessary for Mark V flowfield analysis, but was retained in the current code to ensure no program capability was lost.

### Auxiliary Programs Control Card (10I1)

Column	Variable	Routine Format	Explanation
1	IAUX (1)	AUXILI	Auxiliary program options in the order that they are to be executed. <u>IAUX</u> Option.
2	IAUX (2)	II	
3	IAUX (3)		
.	.		
.	.		
etc.	etc.		= 1 Generate Stability and Control Output File = 2 Not used. Available for user additions. = 3 Not used. = 4 General Cutting Plane Option.
10	IAUX (10)		

### A5.1 Stability And Control Output

This option allows the user to print stability and control data to a file for later use in the Hypervelocity Stability and Control post-processor, POSTSC (References 1 and 2). POSTSC performs trim analysis and calculates stability and control parameters such as short period damping ratio and Dutch roll frequency.

POSTSC requires complete configuration data at a minimum of three control deflections plus static, dynamic and control derivative data. For a stability and control analysis, derivative data is required at zero pitch control deflection. The recommended procedure for generating data to be written to the stability and control output file is given below.

1. Case 1: Calculate all derivatives (IDERIV=9) for the complete configuration (i.e. run all components of the configuration). The pitch control surfaces should be run at all control deflections to be analyzed in POSTSC. Create summations which calculate the aerodynamics of the entire configuration, one summation per control deflection. The summation numbers will then be input for the variable ISET.
2. Case 2: Calculate the pitch control derivatives (IDERIV=2) on only the pitch control component(s), one run per control deflection. These runs are then used in conjunction with the variable IPDP. If the control surface is made up of more than one component (or several control surfaces are used simultaneously), a summation may be used (remember to set the variable IPDP to the summation set number, see note (d) below).
3. Case 3: Calculate the roll control derivatives (IDERIV=5) on only the roll control component(s), one run per control deflection. These runs are then used in conjunction with the variable IPDA. If the control surface is made up of more than one component (or several control surfaces are used simultaneously), a summation may be used (remember to set the variable IPDA to the summation set number, see note (d) below).
4. Case 4: Calculate the yaw control derivatives (IDERIV=5) on only the yaw control component(s), one run per control deflection. These runs are then used in conjunction with the variable IPDR. If the control surface is made up of more than one component (or several control surfaces are used simultaneously), a summation may be used (remember to set the variable IPDR to the summation set number, see note (d) below).

- Note:
- (a) The flight conditions, including  $\alpha$ - $\beta$  set, must be the same for all cases.
  - (b) If the VECC GUI was used to generate cases 1 through 4, the user **must** change the variable IFSAVE to 1 (inviscid pressure method *Title Card and Basic Flags*, page A-66, and viscous analysis if used on the *Skin Friction Basic Flag and Title Card*, page A-74).
  - (c) Remember all runs are counted sequentially on the Force and Moment Data Save unit. Therefore if 10 components are analyzed in Case 1, the first run of Case 2 is Data Set 11 and IPDP=11.
  - (d) Summations are numbered sequentially and independently of the runs. Thus the first summation of Case 1 is Number 1 (ISET=1).

Stability and Control Data Transfer Card (I1,4I2,2X,F10.0)

Column	Variable	Routine Format	Explanation
1	LAST	WRITSC I1	Last control card for data transfer = 0 This is not the last control card = 1 This is the last control card
2-3	ISET	WRITSC I2	Summation set number on the force and moment data storage unit (9) that contains the static and dynamic derivatives for the complete configuration generated with IDERIV=9.
4-5	IPDP	WRITSC I2	Data set number on the force and moment data storage unit (9) that contains the pitch control derivatives generated with IDERIV=2
6-7	IPDA	WRITSC I2	Data set number on the force and moment data storage unit (9) that contains the roll control derivatives generated with IDERIV=5
8-9	IPDR	WRITSC I2	Data set number on the force and moment data storage unit (9) that contains the yaw control derivatives generated with IDERIV=5
12-21	DELTA	WRITSC F10.0	Pitch control deflection for data set ISET

Note: ISET must be input for all data sets written. IPDP, IPDA, and IPDR do not have to be input (may be set to zero).

**Example:**

Consider the case of a configuration with following components:

Nose  
 Body  
 Wing  
 Elevon (both pitch and roll control)  
 Tail (vertical tail used for yaw control)

Case 1:

IDERIV=9

Run	Save Set	Component
Inviscid	1	Nose
Inviscid	2	Body
Inviscid	3	Wing
Inviscid, $\delta=0^\circ$	4	Elevon
Inviscid, $\delta=0^\circ$	5	Tail
Inviscid, $\delta=-20^\circ$	6	Elevon
Inviscid, $\delta=-10^\circ$	7	Elevon
Inviscid, $\delta=10^\circ$	8	Elevon
Inviscid, $\delta=20^\circ$	9	Elevon

Summation	Save Sets	Description
1	1,2,3,4,5	Full Configuration, $\delta=0^\circ$
2	1,2,3,5,6	Full Configuration, $\delta=-20^\circ$
3	1,2,3,5,7	Full Configuration, $\delta=-10^\circ$
4	1,2,3,5,8	Full Configuration, $\delta=10^\circ$
5	1,2,3,5,9	Full Configuration, $\delta=20^\circ$

Case 2:

IDERIV=2

Run	Save Set	Component
Inviscid, $\delta=0^\circ$	10	Elevon
Inviscid, $\delta=-20^\circ$	11	Elevon
Inviscid, $\delta=-10^\circ$	12	Elevon
Inviscid, $\delta=10^\circ$	13	Elevon
Inviscid, $\delta=20^\circ$	14	Elevon

Note: Elevon set as a pitch control surface (IGTYPE=3) on the *Component Organization Card* (page A-66)

Case 3:

IDERIV=5

Run	Save Set	Component
Inviscid, $\delta=0^\circ$	15	Elevon
Inviscid, $\delta=-10^\circ$	16	Elevon
Inviscid, $\delta=10^\circ$	17	Elevon

Note: Elevon set as a yaw/roll control surface (IGTYPE=-3) on the *Component Organization Card* (page A-66)

Case 4:

IDERIV=5

Run	Save Set	Component
Inviscid, $\delta=0^\circ$	18	Tail
Inviscid, $\delta=-20^\circ$	19	Tail
Inviscid, $\delta=20^\circ$	20	Tail

Note: Tail set as a yaw/roll control surface (IGTYPE=-3) on the *Component Organization Card* (page A-66)

The *Stability and Control Data Transfer Card* then looks like the following:

LAST	ISSET	IPDP	IPDA	IPDR	DELTA
0	1	10	15	18	0
0	2	11		19	-20
0	3	12	16		-10
0	4	13	17		10
1	5	14		20	20

## A5.2 General Cutting Plan Option

The general cutting plane option has the capability to determine the section shape of an arbitrary body in any desired plane. That is, the intersection of an arbitrary body cut by a plane. The orientation of the cutting plane is general, being specified by a point in space and three rotation angles. This option allows the user to output points at the intersection of cutting planes with S/HABP panels which can be useful for shock shape/flowfield analysis (see Section 4.3.2.2 of this User's Manual). The intersections which form the section shape are collected for each cutting plane as they are calculated. They may then be collected by: individual panel, all panels as a group, and by panels as sets. Points may be ordered with respect to any of the three body coordinates (X,Y,Z) and also with respect to the axial coordinate (A) of the cutting plane.

### Configuration Description Card (15A4)

Column	Variable	Routine Format	Explanation
1-60	CTITLE	GENCUT 15A4	Configuration Title

### Component Control Card (A4,25XI1,4XI1,3XI2,4XI1,9XI1)

Column	Variable	Routine Format	Explanation
1-4	TITLE	GENCUT A4	Component Identification
30	IORG	GENCUT I1	Origin card flag = 0 Zero (0.0) will be assumed for all origin variables. The <i>Cutting Plane Origin and Orientation</i> card will not be input. = 1 The Cutting Plan Origin and Orientation card will be input.
35	INPHI	GENCUT I1	Cutting Plane Type Flag = 0 Meridian cuts will be equally spaced. = 1 Meridian cut positions will be input = 2 Parallel equally spaced cuts will be generated by the program. Two <i>Parallel Planes Input Position</i> cards will be input to give the position of the first and last cutting planes. = 3 Parallel cuts with arbitrary input spacing will be used. The number of cuts will be equal to NPL and the number of <i>Parallel Planes Input Position Cards</i> will also be equal to NPL.
39-40	NPL	GENCUT I2	Number of cutting planes ( $\leq 36$ )
45	ISYM	GENCUT I1	Symmetry override flag. = 0 Symmetry of the geometry data is ignored = 1 Symmetry of the geometry data is used
55	IPRNT	GENCUT I1	Special print flag = 0 Do not print detailed checkout data = 1 Print detailed checkout data

Cutting Plane Origin and Orientation Card (6F10.0)

Input only if IORG=1

Column	Variable	Routine Format	Explanation
1-10	XP0	GENCUT F10.0	X0 (scaled coordinates)
11-20	YP0	GENCUT F10.0	Y0
21-30	ZP0	GENCUT F10.0	Z0
31-40	PSI0	GENCUT F10.0	PSI0, $\Psi_0$ , degrees
41-50	THET0	GENCUT F10.0	THET0, $\theta_0$ , degrees
51-60	PHI0	GENCUT F10.0	PHI0, $\phi_0$ , degrees

Meridian Planes Input Position Cards (6F10.0)

This card is input only if INPHI = 1. Input 6 points per card. The number of points is equal to NPL.

Column	Variable	Routine Format	Explanation
1-10	PHI (1)	GENCUT 6F10.0	Meridian angles in degrees. Must be in ascending order.
11-20	PHI (2)		
21-30	PHI (3)		
31-40	PHI (4)		
41-50	PHI (5)		
51-60	PHI (6)		

NOTE: Use more cards until PHI (NPL) is reached.

Parallel Planes PHI Card (1F10.0)

This card is input only if INPHI = 2 or 3

Column	Variable	Routine Format	Explanation
1-10	PHICD	GENCUT F10.0	Plane angle in degrees. (= 0.0 for parallel streamwise cuts on a wing).

Parallel Planes Input Position Cards (3F10.0)

These cards are input only if INPHI = 2 or 3.

If INPHI = 2 then two of these cards will be read.

If INPHI = 3 then the number of these cards must be equal NPL

Column	Variable	Routine Format	Explanation
1-10 11-20 21-30	XN YN ZN	GENCUT 3F10.0	The coordinates in the reference coordinate system (body coordinates) through which the cutting plane will pass.

Panel Identification and Control Card (A4,50XI1,5XI1)

Column	Variable	Routine Format	Explanation
1-4	IPANL	GENCUT I4	Panel number. Geometry data on unit 4 corresponding to this number is used.
55	IPRINT	GENCUT I1	Special print flag (check out only)
61	LAST	GENCUT I1	= 0 another card of this type will be read = 1 this is the last panel ID card

A maximum of 10 panel ID cards may be used.

Output Data Control Card (10I1,3XI2,5XA4,5XI1,24XI1)

Column	Variable	Routine Format	Explanation
1 2 3 4 . 10	IOR(1) IOR(2) IOR(3) IOR(4) . IOR(10)	OUTD 10I1	Ordering flag for each panel (input in <i>Panel Identification and Control Card</i> ) individually = 0 no ordering ≠ 0 points will be ordered as per variable flag IOV
14-15	IOG	OUTD I2	Group ordering flag < 0 all panels ordered as if random set = 0 no group ordering = 1 panels ordered by groups
21-24	COMPB	OUTD A4	Base panel of group
30	IOV	OUTD I1	Flag to signify variable to be ordered = 0 axial coordinate (also default) = 1 X-coordinate = 2 Y-coordinate = 3 Z-coordinate
55	IPRINT	OUTD I1	Output print control flag = 0 Do not print results = 1 Print output data



## APPENDIX A REFERENCES

1. Ely, W.L., Moore, M.E., "Stability and Control Of Hypervelocity Vehicles, Volume 1: Technical Report," AFWAL-TR-88-3076, July 1988.
2. Moore, M.E., et. al., "Program User's Manual For: Stability and Control Of Hypervelocity Vehicles, Volume II - Post-Processor (POSTSC, PLOTSC, and PLOTM), " AFWAL-TR-88-3037, December 1988.

*Supersonic/Hypersonic Arbitrary Body Program*

*Mark V Release*

*Appendix B, PREPMK5 (Text Preprocessor) Instructions*

## TABLE OF CONTENTS

Section	Page
B-1. INTRODUCTION .....	B-1
B-2. TEXT PREPROCESSOR DESCRIPTION .....	B-3
B-3. TEXT PREPROCESSOR vs GRAPHICAL USER INTERFACE.....	B-5

## B-1. INTRODUCTION

Prepmk5 is a text preprocessor for the Mark V release of the Supersonic/Hypersonic Arbitrary Body Program (S/HABP). Prepmk5 uses text based menus which allows a user to setup and execute S/HABP from a text only terminal. Prepmk5 follows the input scheme defined by the individual input cards as described in Appendix A, Input Data Instructions. Prepmk5 is fully compatible with the VECC Graphical User Interface (GUI) with the exception of several S/HABP options that are included in the text preprocessor that are not supported by the VECC GUI. The general flow of the program is illustrated in Figure B-1, which shows the major program options.

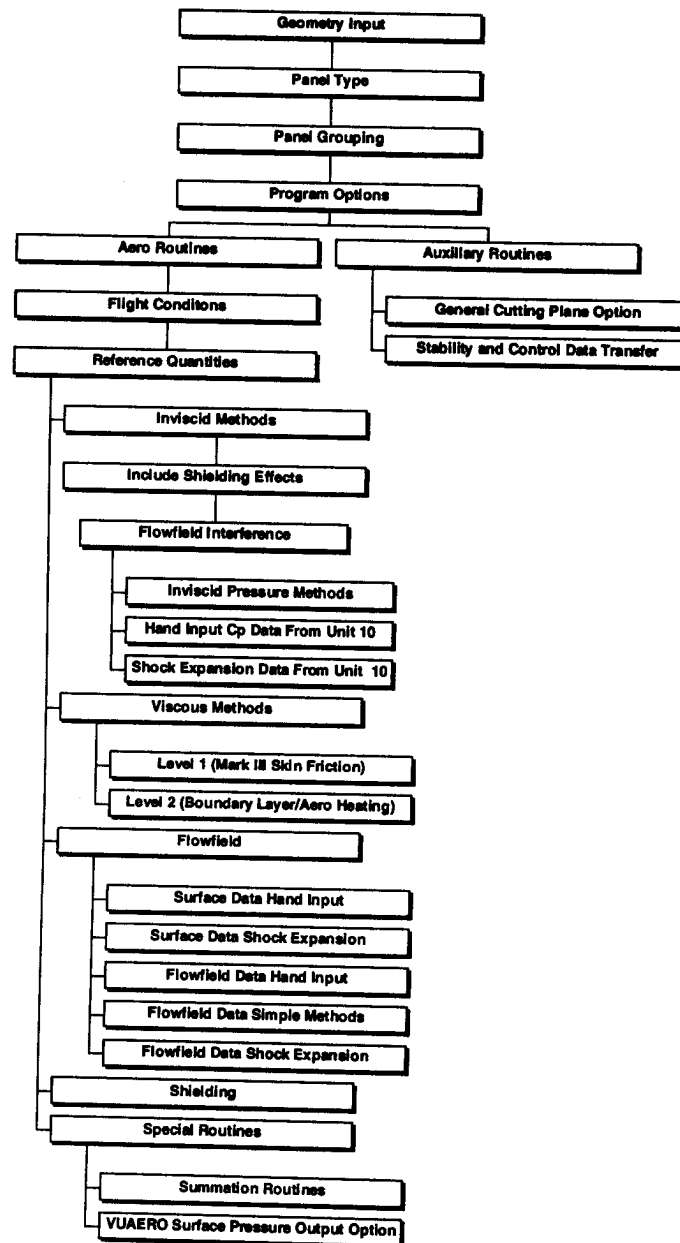


Figure B-1 Prepmk5 Program Flow Diagram

## B-2. TEXT PREPROCESSOR DESCRIPTION

The text preprocessor uses text based menus to setup a S/HABP input file. The text menus used to input data allow for default values, error checking and special options.

The text preprocessor walks the user through the required inputs to define a S/HABP run; however, the user must make sure that the runs are entered in the correct order to ensure proper execution. For example, if the user wants to calculate the inviscid forces on a wing using flowfield interference, the user must first input the flowfield run and then input the inviscid run. For more information on the order of inputs, see Appendix A, Input Data Instructions. A feature of the text preprocessor is that it will only prompt the user for the required inputs and will tailor the menu for any previous data input. For example, if the user is setting up an inviscid run, and selects modified Newtonian for the impact pressure method, the user is only prompted for PDATA(1), PDATA(2), and PDATA(4). The resulting menu for PDATA(1) is customized to reflect the input for the impact pressure method.

The text preprocessor uses several different types of text menus to enter data which allows for default values, limited error checking and special options. There are four basic types of text menus: option menu, numerical input menu, text input menu, and logical menu. There are default values, error checking and special options for all of the menu types except the text input menus which simply take a character string. The default value is either enclosed in square brackets or is capitalized, and may be chosen by simply hitting the carriage return. Error checking consists of checking the input value against a pre-determined range. If the input value exceeds the set range, a warning message is written which gives the minimum and maximum allowable values and returns control to the current menu. Two special options have been incorporated into the menus. The first option, is the capability to exit the program by typing "exit" or "quit" at the prompt. The second option, is the capability to move to the previous menu by typing "up" at any prompt. The second option is only valid for moving around on a single input card. Once a card has been fully input, the user cannot return to the card.

The option menu as shown in Figure B-2, allows the user to select a value from a list of options. The default value is shown in square brackets. If the user attempts to input a number outside the range of the menu, the program outputs an error message and returns control to the current menu. The user can use both special options at this type of menu.

```
ENTER PROGRAM OPTION - IPG [0]
      = 0      NO MORE OPTIONS, EXIT PROGRAM
      = 1      AERODYNAMIC CALCULATIONS
      = 2      AUXILIARY ROUTINES
ENTER CHOICE [ 1]
```

**Figure B-2 Example Option Menu**

A numerical input menu, such as the one shown in Figure B-3, allows the user to enter a numerical value. The default value is shown in square brackets and can be selected by hitting the carriage return. If the user attempts to input a number outside the pre-determined range, the program outputs an error message and returns control to the current menu. Since the range of input values for this type of menu can be varied, the pre-determined range is usually very large. The user can use both special options at this type of menu.

```
FREE STREAM MACH NUMBER - MACH
ENTER VALUE [      5.000]
```

**Figure B-3 Example Numerical Input Menu**

The text input menu as shown in Figure B-4, allows the user to enter a text string. If the user does not enter a text string and hits the carriage return, no string is recorded and execution continues. If the user enters a string that is longer than the allowable length, the string is truncated. The user cannot use either special option at this type of menu.

```
CONFIGURATION IDENTIFICATION (60 CHARACTERS MAXIMUM)
```

**Figure B-4 Example Text Input Menu**

The logical Yes/No menu as shown in Figure B-5, allows the user to answer "yes" or "no" to the question. The default value is shown by a capital letter. The program only compares the first letter of the response and is not case sensitive. If the user attempts to input something other than the appropriate response, the program uses the default value. The "up" special option may be used at this type of menu.

```
WILL THE ANGLE OF ATTACK BE HELD CONSTANT [Y/N] ?
```

**Figure B-5 Example Logical Yes/No Menu**

The text preprocessor incorporates a useful feature that allows the user to enter a constant value for several inputs instead of forcing the user to input the value multiple times. If a variable such as angle of attack has to be input several times, Prepmk5 asks if the input is constant as illustrated in Figure B-6. In the example shown in Figure B-6, if the user responds that the angle of attack will be held constant, the user is only prompted once for a single angle of attack. If the user responds that the angle of attack will not be constant, the user is prompted for each angle of attack required.

```
WILL THE ANGLE OF ATTACK BE HELD CONSTANT [Y/N] ? Y
```

```
VEHICLE ANGLE OF ATTACK - ALPHA (DEG)
ENTER VALUE [      0.000]
```

**Figure B-6 Example Menu Allowing Constant Value Of Multiple Inputs**

### **B-3. TEXT PREPROCESSOR vs GRAPHICAL USER INTERFACE**

The text preprocessor is fully compatible with the VECC GUI; however, it is setup very much differently.

The biggest difference between the text preprocessor and GUI is the structure of the inputs. The text preprocessor puts the burden on the user to enter the runs in the correct order to ensure proper execution. In addition, the text preprocessor steps the user through each input on a specific card. The GUI on the other hand, provides an environment that allows for free input of runs and cases without regard for individual input cards. The GUI frees the user from worrying about ordering the runs during input.

Another difference between the text preprocessor and the GUI is the geometry capabilities. Prepmk5 can read in a standard S/HABP geometry file (Type3 cards) or generate a geometry using the ellipse generation inputs. To make the text preprocessor compatible with the GUI, the text preprocessor generates the appropriate Type3 cards instead of writing the actual ellipse generation cards to the S/HABP input file. The GUI can also read a standard S/HABP geometry file but has more extensive geometry building capabilities.

The text preprocessor supports the AUXILIARY options that are not available in the GUI. The user should be cautioned that these options are not compatible with the GUI and the input file will not be able to be read into VECC.